INTERGRAPH® Smart → 3D

Structural Detailing

User's Guide

PART OF HEXAGON

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Contents

Preface

This document is a user's guide for the Structural Detailing functionality of Intergraph Smart™ 3D and provides command reference information and procedural instructions.

Documentation Comments

For the latest support information for this product, comments or suggestions about this documentation, and documentation updates for supported software versions, please visit *Intergraph Smart Support* (*https://smartsupport.intergraph.com*).

What's New in Structural Detailing

The following changes have been made to the Structural Detailing task.

Version 2016 (11.0)

- Removed the **Place Beam Profile Parts** command. You can still view and edit beam profile parts. (P2 CP:130023)

- Added information about which objects Smart 3D copies as well as what types of modifications are supported by **Copy Symmetry**, **Copy Similar**, and **Copy Across Model**. For more information, see the following:

  Frame mapping and Range Growth Values in Copy by Family (on page 345)
  Objects and behaviors supported for Copy by Family (on page 347)
  Objects not supported by Copy by Family (on page 356)
  Objects and behaviors supported in Attribute Modify (on page 357)
  Solutions to common problems in Copy by Family (on page 364)

  (P3 CP:263679)

- Updated *Objects not supported by Copy by Family* (on page 356), *Objects and behaviors supported in Attribute Modify* (on page 357), and *Solutions to common problems in Copy by Family* (on page 364). (P2 CP:237601, CP:211489, CP:211490, CP:213977, CP:238915, CP:238917, CP:253294)

- Added the **Copy MfgData** option to the **Across Model Family Definition** dialog box. For more information, see *Across Model Family Definition Dialog Box* (on page 335). (P2 CP:245086)

- When you copy a lapped plate using **Delete Optional** in the **Paste** command, the software replaces the lapped plate with a surface body to prevent the plate from going to the **To Do List**. For more information, see *Lapped Plate* (on page 114). (P2 CP:260315)

- Added the **Submit Batch Job** command to run batch processes using the Intergraph Batch Services framework. For more information, see *Submit Batch Job* (on page 383). (P2 CP:273093, P2 CP:178040)

- Added a new locate filter, **Construction Graphics**. For more information, see *Selecting Objects* (on page 49). (P2 CP:112582)

- Updated *Objects and behaviors supported for Copy by Family* (on page 347), *Objects not supported by Copy by Family* (on page 356), and *Objects and behaviors supported in Attribute Modify* (on page 357) for Version 2016. (P3 CP:288254)

- Removed the **Place Profile Straking Seam** command. You can still view and edit profile straking seams. (P3 CP:236319)

- A new command, **Reference Definition Method**, defines how a split point is created. For more information, see *Split Physical Connection* (on page 207). (P3 CP:255918)

- Updated the information related to adding and editing auxiliary parts to an assembly connection. For more information, see *Add auxiliary parts to an assembly connection* (on page 287) and *Edit assembly connection parts* (on page 288). (P3 CP:208142)

- **Import** and **Export** commands and error display are now available for geometric constructions used in **Split Physical Connection**. For more information, see *Geometric Construction Explorer* (on page 273). (P3 CP:185171)

- Added information on the **Pick Edges** option for selecting boundaries. For more information, see *Boundary Methods* (on page 33). (P3 CP:289423)

- Updated the descriptions of the **Thickening Technique Requested** and **Thickening Technique Used** properties on the **General** tab of the **Plate Part Properties** dialog box. For more information, see *General Tab (Plate Part Properties Dialog Box)* (on page 405). (P4 CP:258906)

- Added information on running **Board Management Service** continuously or periodically. For more information, see *Board Management Service* (on page 380). (P4 CP:260909)

Software Changes

- Implemented **Delete Optional** for standalone profile parts. (P1 CP:210957)

- Implemented **Delete Optional** for sketched features. (P1 CP:210991)

- Implemented **Delete Optional** for standalone edge reinforcement parts. (P1 CP:210997)

- Implemented **Delete Optional** for physical connection split points. (P1 CP:210998)

- Implemented **Delete Optional** for assembly connections and children. (P1 CP:211003)

- Implemented **Delete Optional** for free edge treatments. (P1 CP:231346)

- Implemented **Delete Optional** for edge features. (P1 CP:210962)

- Implemented **Delete Optional** for standalone stiffener parts. (P1 CP:257230)

- Implemented **Model Data Transform** for standalone plate parts. (P1 CP:231317)

- Implemented **Model Data Transform** for lapped plate parts. (P1 CP:231318)

- Implemented **Model Data Transform** for standalone bracket parts. (P1 CP:231319)

- Implemented **Model Data Transform** for standalone profile parts. (P1 CP:231320)

- Implemented **Model Data Transform** for edge features. (P1 CP:231321)

- Implemented **Model Data Transform** for sketched features. (P1 CP:231324)

- Implemented **Model Data Transform** for physical connection split points. (P1 CP:231326)

- Implemented **Model Data Transform** for free edge treatments. (P1 CP:231328)

- Implemented **Model Data Transform** for detailed plate parts. (P1 CP:258503)

- Implemented **Model Data Transform** for chamfers. (P1 CP:259090)

- Implemented **Model Data Transform** for standalone edge reinforcement parts. (P1 CP:231325)

- Implemented **Model Data Transform** for assembly connections and children. (P1 CP:231327)

- Implemented **Model Data Transform** for generic member assembly connections. (P1 CP:259089)

- Improved the handling of physicall connections deleted because of no overlapping geometry. (P2 CP:123752)

- Updated batch detailing to use Batch Services. (P2 CP:158942)

- Smart 3D creates welding information between plates, stiffeners, members, and slabs. (P2 CP:176214)

- Updated the algorithm for calculation of plate part length and width. (P2 CP:206583)

- Updated **Lapped Plate** to allow you to select a new lapped to plate after a copy. (P2 CP:260315)

- Added Work Breakdown System (WBS) support for detailing objects. (P2 CP:264164)

- Added support for split none frame connections as input to **Place Assembly Connections**. (P2 CP:264268)

- Improved the stability of physical connection split point after physical connection modifications. (P2 CP:267016)

- Providing the ability to split a physical connection at a distance from an object end without a bounding surface. (P2 CP:247074)

- Support the **Create** and **Delete** steps for standard members, including detailing and manufacturing data. (P2 CP:213978)

  - Support the **Find** step for members. (P2 CP:213972)

  - Support the **Modify** step for standard members. (P2 CP:280054)

- Support the **Create** and **Delete** steps for designed members, including detailing and manufacturing data. (P2 CP:238909)

  - Support the **Modify** step for designed members. (P2 CP:238912)

- Provide **Partial Detailing** options. (P2 CP:280632)

  - Enable SmartPlant License Manager for **Partial Detailing**. (P2 CP:284107)

  - Support **Partial Detailing** in **Copy By Family**. (P2 CP:234390)

- Improve **Copy Across Model** to support detailing and manufacturing. (P2 CP:245086)

Structural Detailing

The Structural Detailing task is used to add details to the basic structure created in Molded Forms. Using this task, you create the following:

Detailed Parts

These parts are created from and replace the root parts created in the Molded Forms task. Detailing uses the parent system's properties, such as thickness, continuity, and molded conventions, to create parts containing complete neat geometry, such as bevels, chamfers, and detailed features.

Standalone Detailed Parts

These parts contain complete neat geometry, but do not have a parent Molded Forms system. These parts are generally for secondary structures.

Assembly Connections

These connections are created automatically as children of logical connections for detailed parts or as the root connection for secondary detailed parts. They contain connections between parts, and maintain relationships between the generated parts resulting from connection rules.

Physical Connections

These connections are created automatically as children of assembly connections. They contain connection properties such as weld or bevel information.

You can start the Structural Detailing task by clicking **Tasks > Structural Detailing**. The Structural Detailing task has these commands:

	Select - Used to select objects in the model. For more information, see *Selecting Objects* (on page 49).
	Execute Detailing - Creates detail parts from the root part of a plate or profile system. For more information, see *Execute Detailing Command* (on page 54).
	Place Plate Parts - Places a standalone detailed planar plate. For more information, see *Place Plate Parts* (on page 92).
	Lapped Plate - Places a standalone lapped plate part. For more information, see *Lapped Plate* (on page 114).
	Place Bracket Parts - Creates a bracket plate part that reinforces other plate parts. For more information, see *Place Bracket Parts* (on page 121).
	Place Stiffener Profile Part by Intersection - Creates a standalone stiffener profile part on the selected detailed plate part by defining the landing curve as the intersection of the selected plate part and a grid plane. For more information, see *Place Stiffener Profile Part by Intersection* (on page 133).
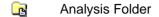
	Place Corner Feature - Places a planar curve from the catalog along a selected edge, at a selected point on the edge, and co-planar to the face of a part. The feature is applied to the part as a cutout. For more information, see *Place Corner Feature* (on page 141).
	Place Multiple Corner Features - Places a planar curve from the catalog at each corner of a selected part. The features are applied to the part as cutouts. For more information, see *Place Multiple Corner Features* (on page 182).
	Place Edge Feature - Places a planar curve from the catalog along a selected edge, at a selected point on the edge, and co-planar to the face of a part. The feature is applied to the part as a cutout. For more information, see *Place Edge Feature* (on page 146).
	Place Edge Features at Seams - Places a planar curve from the catalog where the edge of the part meets a seam. For more information, see *Place Edge Features at Seams* (on page 154).
	Place Edge Features by Offset - Places a series of edge features along an edge at a specified offset distance. For more information, see *Place Edge Features by Offset* (on page 156).
	Place Sketched Features - Places a closed planar curve, either from the catalog or sketched, co-planar to the face of a part. The feature is applied to the part as a cutout, similar to a hole. For more information, see *Place Sketched Features* (on page 160).
	Place Edge Treatment - Specifies the characteristics of free edges. For more information, see *Place Edge Treatment* (on page 167).
	Profile Edge Reinforcement Part - Places a standalone profile part along the free edge of a plate part. For more information, see *Profile Edge Reinforcement Part* (on page 179).
	Plate Edge Reinforcement Part - Places a standalone lapped plate part around a closed edge (such as an opening or sketched feature) of another plate part. For more information, see *Plate Edge Reinforcement Part* (on page 172).
	Straking Seam by Intersection - Creates a straking seam and splits the selected detailed plate part by defining the landing curve as the intersection of the selected plate part and a grid plane. For more information, see *Straking Seam by Intersection* (on page 187).
	Straking Seam by Offset - Creates a straking seam and splits the selected detailed plate part by defining the landing curve as being offset from an existing curve such as another seam, plate boundary, or reference plane. For more information, see *Straking Seam by Offset Command* (on page 190).
	Straking Seam by Projection - Creates a straking seam and splits the selected detailed plate part by projecting a 2D landing curve that you have sketched onto

	the plate part. For more information, see *Straking Seam by Projection Command* (on page 194).
	Straking Seam by Table - Creates a straking seam and splits the selected detailed plate part by defining the landing curve as coordinates in a table. For more information, see *Straking Seam by Table* (on page 198).
	Split Physical Connection - Splits an existing physical connection at a location that you specify. For more information, see *Split Physical Connection* (on page 207).
	Place Member Generic Assembly Connection - Places an assembly connection between a built-up member part and an intersecting part. For more information, see *Place Member Generic Assembly Connection* (on page 275).
	Member Assembly Connections - Places an assembly connection at the selected frame connection. Assembly connections define the necessary trimming between member parts and provides for the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections also control cutbacks, copes, notches, bolt holes, weld preparations, and slots. For more information, see *Place Assembly Connection* (on page 279).
	Place Manual Physical Connection - Creates a physical connection between two parts. For more information, see *Place Manual Physical Connections* (on page 295).
	Place Manual Chamfer - Creates a manual chamfer between two detailed parts. For more information, see *Place Manual Chamfer* (on page 377).
	Board Management Service - Determines structural part and seam symmetry about the center plane. This command is available on the **Tools** menu. For more information, see *Board Management Service* (on page 380).
	Delay Settings - Specifies whether changes made in Molded Forms are immediately updated in Structural Detailing. This command is available on the **Tools** menu. For more information, see *Delay Settings (Tools Menu)* (on page 387).
	Submit Batch Job - Runs batch processes using the Intergraph Batch Services framework. This command is available on the **Tools** menu. For more information, see *Submit Batch Job* (on page 383).
	Check Manufacturability - Analyzes pipes in the current workspace and reports the objects that will be difficult or impossible to manufacture. This command is available on the **Tools** menu. For more information, see *Check Manufacturability* (on page 392).

See Also

Structural Detailing Workflow (on page 27)

Icons in the Workspace Explorer

Analysis Folder

Analysis Model

Assembly

Assembly Connection

Beam Part - undetailed

Beam Part - detailed

Beam Part - detailed, manufactured

Bearing Plate - undetailed

Bearing Plate - detailed

Bearing Plate - detailed, manufactured

Bearing Plate - group member

Bearing Plate - group member, manufactured

Bearing Plate - group master

Bearing Plate - group master, manufactured

Block

Boundary Condition

Bracket System or Leaf System

Bracket Part - undetailed

Bracket Part - detailed

Bracket Part - detailed, manufactured
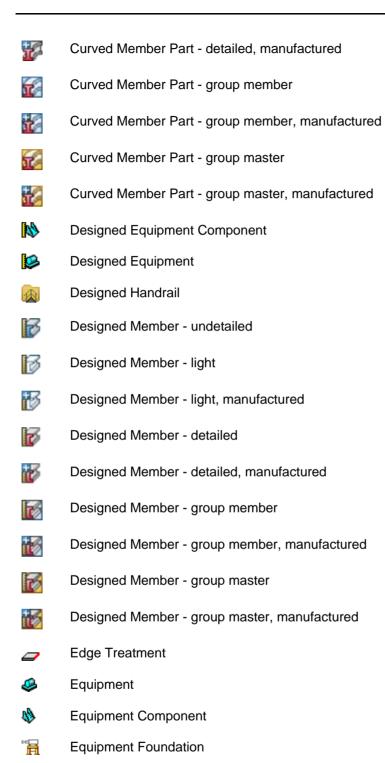
Bracket Part - group member

Bracket Part - group member, manufactured

Bracket Part - group master

Bracket Part - group master, manufactured

Cable Nozzle on Equipment

Cabletray Nozzle on Equipment

Collar Plate - undetailed

Collar Plate - detailed

Collar Plate - detailed, manufactured

Collar Plate - group member

Collar Plate - group member, manufactured

Collar Plate - group master

Collar Plate - group master, manufactured

Common Part Manager

Conduit Nozzle on Equipment

Coordinate System

Coordinate System Axis

Coordinate System Grid Plane

Coordinate System Elevation Plane

Coordinate System Radial Plane

Coordinate System Radial Cylinder

Curved Member Part - undetailed

Curved Member Part - light

Curved Member Part - light, manufactured

Curved Member Part - detailed

Curved Member Part - detailed, manufactured

Curved Member Part - group member

Curved Member Part - group member, manufactured

Curved Member Part - group master

Curved Member Part - group master, manufactured

Designed Equipment Component

Designed Equipment

Designed Handrail

Designed Member - undetailed

Designed Member - light

Designed Member - light, manufactured

Designed Member - detailed

Designed Member - detailed, manufactured

Designed Member - group member

Designed Member - group member, manufactured

Designed Member - group master

Designed Member - group master, manufactured

Edge Treatment

Equipment

Equipment Component

Equipment Foundation

Equipment Shape

Equipment Solid

Equipment Solid - Added

Equipment Solid - Subtracted

Equipment Solid - Suppressed

Feature (includes corner feature, edge feature, face feature, slot, and so on)

Footing

Footing Components

Foundation Port

Frame Connection

Handrail

HVAC Nozzle on equipment

Insert Plate - undetailed

Insert Plate - detailed

Insert Plate - detailed, manufactured

Insert Plate - group member

Insert Plate - group member, manufactured

Insert Plate - group master

Insert Plate - group master, manufactured

Ladder

Linear Member Part - undetailed

Linear Member Part - light

Linear Member Part - light, manufactured

Linear Member Part - detailed

Linear Member Part - detailed, manufactured

Linear Member Part - group member

Linear Member Part - group member, manufactured

Linear Member Part - group master

Linear Member Part - group master, manufactured

Load Case

Load Combination

Member Assembly Connection Cutback Feature

Member Assembly Connection Plate Part

Member Assembly Connection

Member Fireproofing

Member Part

Member Split Connection

Member System

Opening

Panel - manufactured

Physical Connection

Pipe Nozzle on Equipment

Planning Joint - butt weld

Planning Joint - lap weld

Planning Joint - tee weld

Planning Joint Folder

Plate - light

Plate Part - undetailed

Plate Part - light

	Plate Part - light, manufactured
	Plate Part - detailed
	Plate Part - detailed, manufactured
	Plate Part - detailed, manufactured, with manufacturing part assembly child
	Plate Part - group member
	Plate Part - group member, manufactured
	Plate Part - group master
	Plate Part - group master, manufactured
	Plate System or leaf system
	Profile Edge Reinforcement System or Leaf System
	Profile Edge Reinforcement Part
	Profile Edge Reinforcement Part - light
	Profile Edge Reinforcement Part - light, manufactured
	Profile Edge Reinforcement Part - detailed
	Profile Edge Reinforcement Part detailed, manufactured
	Profile Edge Reinforcement Part - group member
	Profile Edge Reinforcement Part - group member, manufactured
	Profile Edge Reinforcement Part - group master
	Profile Edge Reinforcement Part - group master, manufactured
	Seam - Design
	Seam - Intersection
	Seam - Planning
	Seam - Straking

Slab Assembly Connection

Slab

Stair

Standalone Beam Part - undetailed

Standalone Beam Part - detailed

Standalone Beam Part - detailed, manufactured

Standalone Plate Part - undetailed

Standalone Plate Part - detailed

Standalone Plate Part - detailed, manufactured

Standalone Plate Part - group member

Standalone Plate Part - group member, manufactured

Standalone Plate Part - group master

Standalone Plate Part - group master, manufactured

Standalone Profile Edge Reinforcement Part - undetailed

Standalone Profile Edge Reinforcement Part - detailed

Standalone Profile Edge Reinforcement Part - detailed, manufactured

Standalone Profile Edge Reinforcement Part - group member

Standalone Profile Edge Reinforcement Part - group member, manufactured

Standalone Profile Edge Reinforcement Part - group master

Standalone Profile Edge Reinforcement Part - group master, manufactured

Standalone Stiffener Part - undetailed

Standalone Stiffener Part - detailed

	Standalone Stiffener Part - detailed, manufactured
	Standalone Stiffener Part - group member
	Standalone Stiffener Part - group member, manufactured
	Standalone Stiffener Part - group master
	Standalone Stiffener Part - group master, manufactured
	Stiffener Part - undetailed
	Stiffener Part - light
	Stiffener Part - light, manufactured
	Stiffener Part - detailed
	Stiffener Part - detailed, manufactured
	Stiffener Part - group member
	Stiffener Part - group member, manufactured
	Stiffener Part - group master
	Stiffener Part - group master, manufactured
	Stiffener System or Leaf System
	Wall Assembly Connection
	Wall Part
	Wall Run
	Wall System

Structural Detailing Workflow

All structural detailing objects are placed in the model using information inherited from molded forms systems or defined in the structural detailing reference data. The first step should be to review, edit, and otherwise customize the delivered structural detailing reference data. This step is usually performed by an administrator or specialized user before modeling starts.

Structural detailing occurs after structural design has been completed or has progressed to a predefined stage. In the software, structural detailing is performed on molded forms systems, so molded forms objects must be placed first.

Execute Detailing on Systems

When you detail plate and profile systems, the software replaces root parts with detailed parts. As parts are detailed, additional detailing objects are automatically created, such as assembly connections, physical connections, plate chamfers, profile end cuts, and slots and collars for profiles penetrating plates. These objects are defined as smart occurrences by rules in the reference data and can be individually modified after creation.

Modify Smart Occurrence Objects

Optionally, after parts have been detailed, smart occurrence objects can be modified by changing values on Properties dialog boxes:

- *Assembly Connection Properties Dialog Box* (on page 410)
- *Slot Properties Dialog Box* (on page 412)
- *Collar Properties Dialog Box* (on page 414)
- *Free End Cut Properties Dialog Box* (on page 418)
- *Web Cut Properties Dialog Box* (on page 420)
- *Flange Cut Properties Dialog Box* (on page 422)
- *Physical Connection Properties Dialog Box* (on page 424)
- *Chamfer Properties Dialog Box* (on page 432)

Place Secondary Parts

Create standalone plate and profile parts. In general, these parts are of secondary strength importance and include objects such as headers, brackets, and doublers. You can place several types of secondary parts:

- *Place Plate Parts* (on page 92)
- *Lapped Plate* (on page 114)
- *Place Bracket Parts* (on page 121)
- *Place Stiffener Profile Part by Intersection* (on page 133)

Place Features

After parts have been detailed, add features. You can place several types of features:

- *Place Corner Feature* (on page 141)
- *Place Edge Feature* (on page 146)
- *Place Sketched Features* (on page 160)

Place Straking Seams

Place straking seams as needed. Straking seams are used to break plate parts into practical sizes, restricted by what can be purchased, work center limitations, or other fabrication considerations. You have several methods for placing seams:

- *Straking Seam by Intersection* (on page 187)
- *Straking Seam by Offset Command* (on page 190)
- *Straking Seam by Projection Command* (on page 194)
- *Straking Seam by Table* (on page 198)

Place Edge Reinforcement

Place edge reinforcement parts as needed. You have two methods for placing edge reinforcement parts:

- *Plate Edge Reinforcement Part* (on page 172)
- *Profile Edge Reinforcement Part* (on page 179)

Copy in Structural Detailing

📝 **NOTE** You can use the data created in Structural Detailing with Model Data Reuse and Model Data Transform, including the **Delete Optional** functionality.

See Also
Structural Detailing (on page 16)

Plane Methods

You must often define planes used for object surfaces and sketching planes. Usually, but not necessarily, these planes are based from an existing plate or from a grid or reference plane created using the Grids task. The following methods define planes:

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector

Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

What do you want to do?

- *Define a coincident plane* (on page 29)
- *Define an offset plane* (on page 30)
- *Define a plane using a point and a normal vector* (on page 32)
- *Define a plane using three points* (on page 31)
- Move a defined plane
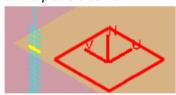
Define plane using angle from plane

1. Click **Angle from Plane** .

2. In the model or **Workspace Explorer**, select the base plane from which to angle the plane that you are defining. This plane can be a planar plate system, face port of a plate port, or a grid plane.

3. Select a rotation axis. The rotation axis must be parallel to the surface of the selected plane.

    TIP The rotation axis can be a linear seam, profile system, landing curve, system edge or connection.

4. In the **Angle** box, specify the rotation angle of the plane relative to the selected base plane.

Define a coincident plane

1. Click **Offset from a Plane** .

2. In the model or **Workspace Explorer**, select a reference plane, a grid plane, or a planar plate system.
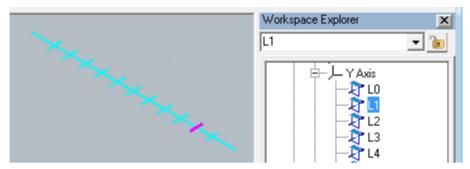
    A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.
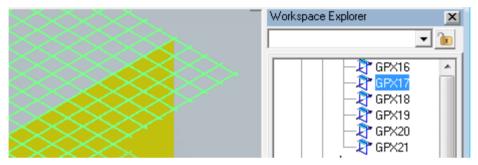
3. Check that **Offset Lock** is locked and that the **Offset** value is 0.

NOTES

- You create reference planes in the Grids task by selecting **Ship** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can select reference planes in the **Workspace Explorer** or in a graphic view.

- You create grid planes in the Grids task by selecting **Grids** in the **Grid Type** box of the **Grid Wizard**. For this plane method, you can best select grid planes in the **Workspace Explorer**.
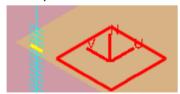
- For more information, see *Create Coordinate System (Grid Wizard)* in the *Grids User's Guide*.

Define an offset plane

1. Click **Offset from a Plane** .

2. In the model or **Workspace Explorer**, select the base plane from which to offset the new plane. The base plane can be a reference plane, a grid plane, or a planar plate system.

   A preview of the selected plane displays. The local coordinate system of the plane displays at the plane's center.

3. In the **Offset** box, type the offset distance from the plane and press TAB.

   The plane moves to the new location.

   -OR-

4. Graphically define the offset:

      a. Click **Offset Lock** 🔒.

      The option changes to unlocked 🔓*.*

      b. Move the pointer to the required location and click.

      The plane and plane coordinate system move to the new location, **Offset Lock** 🔓
      changes to locked 🔒*, and the offset value displays in the* **Offset** *box.*

      c. If an adjustment to the offset is needed, type a new value in the **Offset** box.

5. To move the plane independent of its previous definition, see Move a defined plane.

💡 TIPS
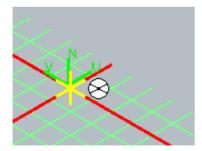
- You can graphically define the offset with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.

- By moving the pointer back and forth over the base plane in the graphic view, you can change the offset direction. You can also change the direction by changing the sign in the **Offset** box.

- When **Offset Lock** is set to locked 🔒, the **Offset** value does not change when the pointer moves.

Define a plane using three points

1. Click **Plane by Three Points** 📐.

2. Specify on the first point 📍 that defines the plane.

3. Specify the second point 📍 that defines the plane.

4. Specify the third point 📍 that defines the plane.

📝 NOTES

- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

- You can define points on grid planes, especially at intersections.

You can control the types of points using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.
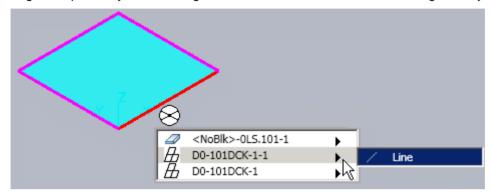
In Molded Forms, define parametric points by using **Insert** > **Topological Points**. For more information, see Topological Points (Insert Menu).
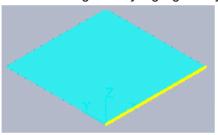
Define a plane using a point and a normal vector

1. Click **Plane by Point and Vector** .

2. Select a vector in the model that is normal to the required plane. The vector can be any linear geometry that is part of the construction of a model object, such as a plate system edge or a profile system landing curve. Use **QuickPick** to select a line geometry element.
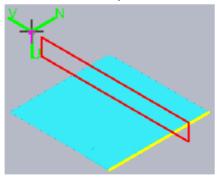
3. Click the **Line** geometry element.

   The selected geometry highlights in yellow.

4. Move the cursor to a point that is on the required plane, and then click.

   A preview of the plane and its local U_V_N coordinate system displays. The plane is coincident with the point and normal to the vector.

5. To move the plane independent of its previous definition, see Move a defined plane.
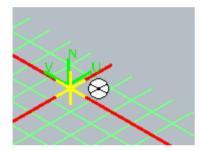
   TIP You can graphically define the point with the best precision by also using **PinPoint**. For more information, see *PinPoint* in the *Common User's Guide*.

Define plane by vectors normal

1. Click **Plane by Vectors Normal** ⬛.

2. In the model or **Workspace Explorer**, click the plane 🗗 to which the new plane is normal.

   💡 **TIP** The plane can be a planar plate system, a face port of a plate part, or a grid plane.

3. Specify the first point 🗗 that defines the vector.

4. Specify the second point 🗗 that defines the vector.

📝 **NOTES**

- You can define points on the surfaces, edges, and corners of systems and parts. You can control the types of points by using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

- You can define points on grid planes, especially at intersections.

  You can control the types of points using **Tools** > **Options**. For more information, see *SmartSketch Tab (Options Dialog Box)* in the *Common User's Guide*.

- In Molded Forms, define parametric points by using **Insert** > **Topological Points**. For more information, see Topological Points (Insert Menu).

Boundary Methods

You must often define boundaries when creating plate systems, profiles, edge reinforcements, and other objects. The common methods for defining boundaries are:

🔲 **Pick Boundaries**

  Define boundaries by selecting them graphically either in a graphics view or in the **Workspace Explorer**.

🔲 **Pick Edges**
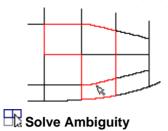
  This option is only available if you pick a plate as a boundary. Select the plate edges to use.

  Click **Pick Boundaries** 🔲 to exit the **Pick Edges** option and select other boundaries.

🔲 **Boundary List**

  Define boundaries by selecting them from a list of potential boundaries. For more information, see *Boundary List Dialog Box* (on page 45).

🔲 **Boundary Curve**

Define the boundary by drawing its shape.

Solve Ambiguity

If the boundaries that you selected can form the object in more than one way, then you have defined an ambiguous solution. The software prompts you to select one or more solutions.

What do you want to do?

- *Pick boundaries* (on page 34)
- *Use the Boundary List dialog box* (on page 34)
- *Define a boundary curve* (on page 35)
- *Solve ambiguous solution created by selected boundaries* (on page 35)
- Rebound objects

Pick boundaries

1. Click **Pick Boundaries** .

2. In a graphic view or **Workspace Explorer**, select an object or grid plane to use as a boundary.

3. Continue selecting objects or grid planes until you have completed defining the boundaries for the object.

NOTE To remove an object from the group of boundaries, select the object again.

Use the Boundary List dialog box

1. On the command ribbon, click **Boundary List** .

   *The **Boundary List** dialog box displays.*

2. To remove a boundary, select the boundary in the list, and then click **Remove** .

3. To remove all boundaries, click **Remove All** .

4. To add a new boundary, click **Add** , and then select a boundary in the graphic view or in the **Workspace Explorer**. You can also type the boundary name and an optional offset value in the **Offset** box or the **Name** box.

NOTE For more information, see *Boundary List Dialog Box* (on page 45).

Define a boundary curve

1. Select **Boundary Curve** .

2. Define the sketching plane for the curve.

   Define a coincident plane (on page 29)
   Define an offset plane (on page 30)
   Define plane using angle from plane (on page 29)
   Define a plane using a point and a normal vector (on page 32)
   Define a plane using three points (on page 31)
   Define plane by vectors normal (on page 33)

3. Click **Add Intersecting Item** .

4. Select objects in the model that intersect the sketching plane that you want to use as reference geometry. These extra objects appear in the 2D environment.

5. Click **Add Projection Item** .

6. Select objects in the model that do not intersect the sketching plane and that you want to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the 2D environment.

7. Select **Auto** to automatically add all related objects to the select set, if needed.

8. Click **Sketch 2D** .

   The 2D environment appears.

9. In the 2D environment, sketch the boundary.

   NOTE You can constrain the boundary to geometric construction points referenced from profile cross-section key points or offset from plate systems. For more information, see *Point at Offset from Keypoint Command* and *Point at Minimum Distance Command* in the *SmartSketch Drawing Editor Drawing Editor Help*, available within the 2D environment.

10. Click **Finish** in the 2D environment.
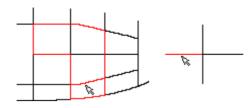
    The 3D environment and the main command ribbon return.

NOTE Deleting a boundary curve from the 2D environment also removes it from the **Boundary List** dialog box.

Solve ambiguous solution created by selected boundaries

The command switches to **Solve Ambiguity** automatically if an ambiguous solution exists after clicking **Accept** or **Finish**. All possible solutions created by the selected boundaries appear in the graphic view.

1. Move the pointer over a solution, and then click to select it.

2. Continue to select solutions until you have defined the entire bounded object. A correct solution to ambiguity meets these conditions:

   - Each solution must have a common edge with at least one other solution.

   - Each boundary must be used by at least one solution.

NOTES

- To modify the ambiguous solution after creating an object, select **Solve Ambiguity** , and select new solutions.

- To remove a previously-selected solution, select the solution again.

- Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Sketch 2D Best Practices

The **Sketch 2D** option is used in several commands to create sketched boundaries, openings, curves for extruded plate systems, and profile and seam landing curves. The following Sketch 2D best practices help you create objects in Molded Forms that you detail in the Structural Detailing task and manufacture in the Structural Manufacturing task.

Templates

Sketch 2D default display options are saved in the file *[Product Folder]*\Common2D\Symbol2D\Templates\template.sha. You can change the default display by changing options in this file:

1. Right-click template.sha and select **Properties**.

2. Clear **Read-only** and click **OK**.

3. Open template.sha and change any of the following default display options:

   - **File** > **Properties**. You can change the units displayed in command ribbon bars on the **Units** tab.

   - **View** > **Toolbars**. You can select the toolbars that are displayed.

   - **Format** > **Dimensions**. You can change the units used in dimensions, the text size, and other dimension properties.

   - **Tools** > **Options**. You can change the default colors on the **Colors** tab.

   - **Tools** > **Display Manager**. You can change the default geometry color, line type, and width for individual layers.

   - **Tools** > **SmartSketch Settings**. You can select the SmartSketch relationships that are enabled on the **Relationships** tab.

4. Save and close the file.

Fractional Units

The default units and precision values in Sketch 2D are defined in a 3D task. For example, if the file *[Product Folder]*\Common2D\Symbol2D\Templates\template.sha has default units set to

mm, and the units of measure in Molded Forms are set to **ft-in**, Sketch 2D defines the units of measurement as **ft-in**.

The **Properties** dialog box in Sketch 2D allows you to view the defined units of measurement and to specify the fractional precision value. If you need to use a specific fractional precision value for all tasks in Sketch 2D, you can specify the precision value in the template file, which takes precedence over the precision value defined by the 3D task.

The following example shows how to modify the length readout precision to **1/64** in the template file:

1. In the template.sha file, set the precision value to **1/64**, and save the file.

2. In the Molded Forms task, click **Tools > Options**.

    *The **Options** dialog box displays.*

3. Under **Units of Measure**, set the distance readout to **ft-in (fractional)**.

4. Click **OK**.

    *The distance units in Sketch 2D are now in **ft-in** with a precision value of **1/64**.*

Number of sketch files allowed per plate

Sketch 2D geometry for an object is saved in 2D files cached in the model and related to the object.

The **Boundary Curve** ✐ step for the plate system commands uses one 2D file. All boundary curves are in one 2D file for each plate system, and are modified, added, or deleted in the same file.

The **Place Opening** ▣, **Stiffener Profile System by 2D Projection** ▨, and **Design Seam by 2D Sketch** ▨ commands can each have:

-   Multiple 2D files, with a separate file for each object on one plate system. The objects are modified in the 2D file, and deleted from the graphic view or **Workspace Explorer**.

-   Geometry for multiple objects in one 2D file. The objects are modified or deleted in the 2D file.

-   One 2D file that applies to multiple plate systems. Changes to the 2D file are reflected on all plate systems.

-   A combination of these files.

Add items to sketch

The **Add Intersecting Item** ▨ and **Add Projection Item** ▧ steps are used to add reference structure to Sketch 2D. Select the minimum number of needed items.

▤ **NOTE** More than 30 items added to a sketch file can slow down software performance when the object is recomputed.

Next, you can select **Auto** to automatically add all objects that are relative to the object to be sketched to the current select set. Items that pass the criteria are added to the select set and are highlighted in the graphic view, as well as the **Workspace Explorer**.

The filter criteria that the software uses to determine eligible candidates depend upon whether **Add Intersecting Item** or **Add Projection Item** was selected in a previous step. The following

table lists the various relevant objects that are automatically imported based on the target, or base sketch, object type.

Base Sketch Object	Intersecting Objects	Projecting Objects
Plate System	▪ Stiffeners ▪ Edge Reinforcements ▪ Seams ▪ Bounded Objects ▪ Bounding Objects ▪ Knuckle Reference Curves	▪ Reference Curves ▪ Stiffeners ▪ Penetration Plates
Plate Part	None supported in the current version of the software	None supported in the current version of the software
Profile System	None supported in the current version of the software	None supported in the current version of the software
Member	None supported in the current version of the software	None supported in the current version of the software
Designed Member (Built Up)	Same as Plate System	Same as Plate System

If no objects are added to the select set, the software displays a message on the status bar.

If you clear **Auto,** all selected objects, whether added to the select set automatically or manually, are cleared. Selecting **Auto** adds back those objects that initially passed the filter criteria, but it does not add back any objects that you selected manually.

⚠ CAUTION When the selected plate system is large, **Auto** automatically adds all objects relative to the sketch in the context of *intersecting* or *project*. This can slow down software performance.

Layers

The software automatically defines several layers in Sketch 2D. Reference items (selected automatically by the software or manually using the **Add Intersecting Item** 🖼 and **Add Projection Item** 🔧 steps) are placed on those layers:

Default Layer

The software only uses geometry that you create on the **Default** layer. You can put other geometry, such as construction lines, on other layers. The Default layer must be the first layer alphabetically. To avoid a conflict, you should add a "z" prefix to any layer which you add, such as "z_construction."

Infinite Elements Layer

Contains reference items with infinite geometry intersecting the sketch plane, such as grid planes. Objects on this layer can be used with dimensions and relationships.

Inputs Layer

Contains other reference items intersecting the sketch plane that can also be used with dimensions and relationships. The reference geometry includes openings, design seams, plate systems, and profile systems.
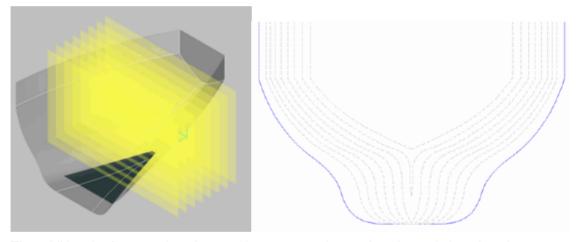
No_Constrained_Elements Layer

Contains reference items that do not intersect the sketch plane. Also contains reference items that cannot be used with relationships and dimensions, such as plate and profile parts. By default, you cannot select reference geometry on the **No_Constrained_Elements** layer. To make the geometry selectable, you must make **No_Constrained_Elements** the active layer, and then switch back to the **Default** layer.

You can add relationships and dimensions to reference geometry on the **No_Constrained_Elements** layer, but the relationships are not saved when you click **Finish**.

Multiple reference planes

You can add reference planes to Sketch 2D that are different than the sketching plane, but parallel to the sketching plane. This is useful in defining landing curves for projected profiles and seams on the hull. Use **Add Projection Item** to add the reference planes.

The additional reference plane intersection geometry is gray in color and placed on the **No_Constrained_Elements** layer.

Using SmartSketch Drawing Editor to save sketch geometry

SmartSketch Drawing Editor, delivered to *[Product Folder]*\Shape2D\Bin\shape2dserver.exe, can be used to save geometry commonly used in Sketch 2D, such as:

- Mother curves and extrusion curves for extruded plate systems.
- Boundary curves.
- Openings or sketched features not included in the Catalog.
- Landing curves for projected profiles and seams.

You can copy the geometry from Sketch 2D and paste it into SmartSketch Drawing Editor, or create the geometry directly in SmartSketch Drawing Editor. Geometry can be saved in multiple

SmartSketch Drawing Editor files or into a single file with multiple sheets. The geometry can be copied from SmartSketch Drawing Editor and pasted into Sketch 2D as needed.

Deleting reference geometry

You can delete reference geometry from the sketch when the software includes geometry that you do not want to see. The reference structure is not deleted, and the reference geometry is recreated the next time that you open the sketch.

Modifying groups of geometry
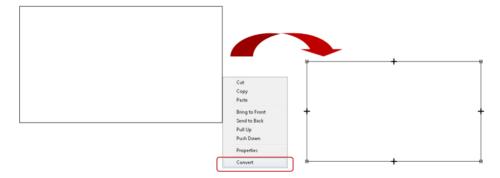
The software automatically groups Sketch 2D geometry when you click **Finish**. When you modify the existing 2D geometry of an object, you must maintain the original group identifier by using the following methods:

1. To modify or delete existing geometry, use **QuickPick**, **Top Down** 🖱 or **Bottom Up** 🖱 with **Select Tool** 🖱 to select the geometry instead of the group.

2. To add new geometry to a group, use **Edit** > **Group Modify**.

3. To replace all geometry in a group with geometry copied to the clipboard, use **Replace Group Elements** 🖱.

Rectangles

Use **Line** ╱ or **Line/Arc Continuous** 🖱 to create rectangles.

If you use **Rectangle** ▢, you must select **Tools** > **Maintain Relationships**. Right-click the geometry, and then select **Convert**. This converts the rectangle into four lines and the correct connection relationships are created at the corners.

Constraints

Use relationships and dimensions to constrain geometry to reference structure. This keeps the geometry properly related to the reference structure if the reference structure changes.
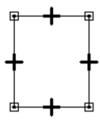
Keep relationships and dimensions as simple as possible. This allows the geometry the best chance to update if the reference structure changes.

Connections
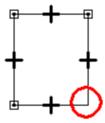
Connection relationships are required for the software to create connected geometry.

Do not create a connection between the corner of the sketched geometry, and the corner of the bounding object. Instead, move the constraint slightly off the corner.
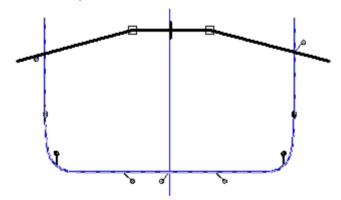
Good: All endpoints of opening are connected.

Bad: Missing connection at one corner causes the creation of the opening to fail.

Good: All endpoints of extrusion curve are connected.

Bad: Missing connection on extrusion curve.
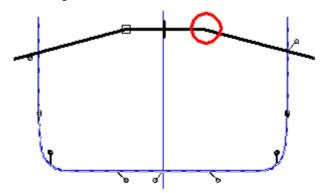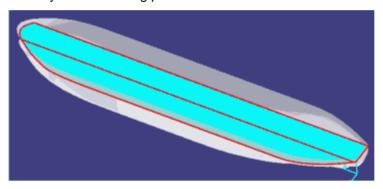
Plate system is missing part of the curve.
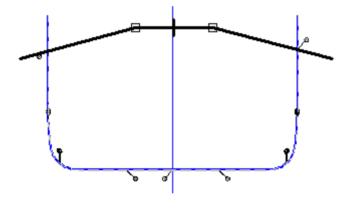
Extending sketch geometry to intersect bounding structure
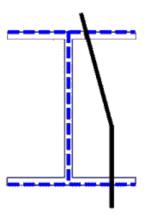
Extend sketch geometry beyond bounding reference structure geometry to ensure that the sketch geometry intersects the reference structure at all locations.

Good:
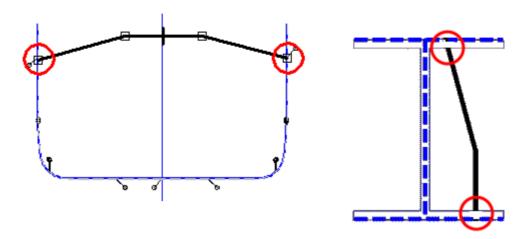
Extrusion curve extends beyond the hull.

Bracket boundary curve extends beyond the top flange and the bottom of the bottom flange.
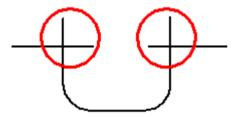
Bad:

Overlapping sketch geometry for boundary curves

A boundary curve with all geometry connected is treated as one edge in the Graphics View. You may sometimes want parts of the curve to be treated as separate edges at sharp discontinuities such as corners. This allows you to add edge reinforcements in the Molded Forms task or edge treatments in the Structural Detailing task to part of the boundary curve without defining additional boundaries. In order to separate the boundary curve into separate edges, overlap the boundary curve geometry instead of connecting it at the corners:
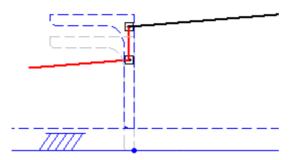
Creating boundary curves on the thickness side of a plate

If your plate boundaries include a plate system with the thickness direction towards the plate that you are creating, a profile mounted to the bounding plate system and a sketched boundary, and then the boundary curve must intersect both the profile system and profile part.

In the following example:

- The solid blue line is the molded surface of the bounding plate system.

- The dashed blue line is the thickness direction for the bounding plate part.

- The dashed gray profile outline is the bounding profile system mounted to the bounding plate system at the molded surface (on the **No_Constrained_Elements** layer).

- The dashed blue profile outline is the bounding profile part mounted to the thickened bounding plate part.

- The solid black line is the intended boundary curve intersecting the profile part.

- The solid red line is the required adjustment to the boundary curve, passing through both the profile part and profile system, and intersecting the profile system.
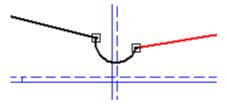
The black line is the intended boundary curve. It can be constrained to the dashed blue profile part using relationships and dimensions. The red lines are additional geometry that must connect to the black boundary curve, pass through the profile part geometry, and intersect the dashed grey profile system geometry.

Arcs in boundary curves
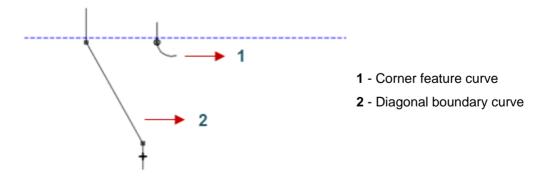
If a boundary curve ends with a circular or elliptical arc, you must add a line segment pointing away from the direction of the arc. This prevents the arc from extending back onto the boundary curve as the curve is calculated.

In the following example, the black curves are the intended boundary curve. The red line is the extension to the boundary curve that prevents the arc from extending back onto the boundary curve.

Multiple curves

Do not create a close boundary curve contour that overlaps trimming input geometry. Instead, draw two separate curves. In the following example, there are two separate curves, one for corner feature and one for diagonal boundary.

1 - Corner feature curve

2 - Diagonal boundary curve

Associative offsets

The **Associative Offset** command allows you to create a curve based on an existing curve, and place the new curve at a specified offset. To use this command, you must add it to a toolbar.

1. Right-click an existing toolbar, and click **Toolbars**.

2. Click **Customize** on the **Toolbars** dialog box.

3. Click **Manipulation** on the **Categories** list.

4. Drag the **Associative Offset** icon from the **Buttons** list to an existing toolbar.

For more information on the **Associative Offset** command see the **Associative Offset Command** topics in the *SmartSketch Drawing Editor* help file*.*

Boundary List Dialog Box

Displays the boundaries for the object that you are placing:

- Model objects, such as plate systems or profile systems.

- Reference planes and other plane definitions.

- Sketched boundary curves created using the **Sketch 2D** option on a supported ribbon. Boundary curves are prefixed with **SketchGroup_.**

- A standalone plane created during a copy, move, or mirror operation when a boundary was not included in the operation. You usually want to replace the standalone plane with a new boundary.

Open this dialog box by clicking **Boundary List** on the command ribbon during the boundary definition step.
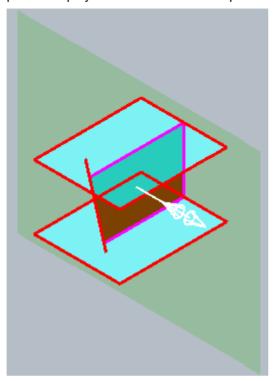
Preview Type

You can press ARROW DOWN ↓ and ARROW UP ↑ to step through the boundary list. Each selected boundary highlights in the graphic view according to the selected preview type. **Tools** > **Options** defines the **Highlight** color. For more information, see *Colors Tab (Options Dialog Box)* in the *Common User's Guide*.
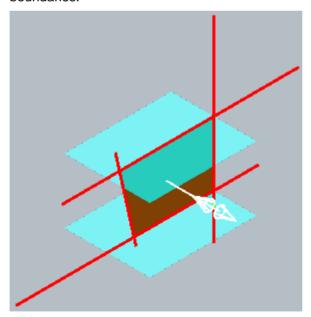
In the following examples, two plate systems, a reference plane, and a sketched boundary are the selected boundaries of the vertical plate.

Bounding Objects - Highlights the selected boundary object as an outline. Reference

planes display as translucent shaded planes.
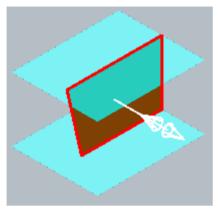
Bounding Geometry - Highlights the internal boundary geometry used by the software. The internal geometry is typically an unlimited length line for model objects and reference plane boundaries.
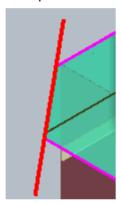
Edges - Highlights edges of the bounded object where the bounded object intersects with

the selected boundary.

NOTE For **Bounding Objects** and **Bounding Geometry**, the complete sketched boundary curve geometry displays as drawn in **Sketch 2D** 🗺, as shown in the following example.

➕ **Add**

Specifies a new boundary. You can select a boundary in the graphic view or the **Workspace Explorer**.

NOTES

- You can also type the boundary name and an optional offset value in the **Name** box or the **Offset** box, and then press TAB or ENTER.
- Invalid text displays with a gray background in the **Name** box or the **Offset** box.

Remove

Removes the selected boundary from the list.

NOTE Deleting a boundary curve from the 2D environment (**Sketch 2D** 🗺 on the main command ribbon) also removes it from the **Boundary List** dialog box.

🧹 **Clear All**

Removes all boundaries from the list.

/ Disconnect

Removes the relationships of a boundary to its definition method. When you disconnect a boundary, it becomes a standalone plane. This option is active by default for a standalone plane created during a copy, move, or mirror operation.

NOTE After disconnecting a boundary, you must select a replacement boundary in the graphic view or the **Workspace Explorer**.

Extend Sketch Boundary in Graphic View

Extends a sketched boundary to a point in space or to another element selected in the graphic view. This option is available when you select a sketched boundary (with a name prefixed with **SketchGroup_**).

Name

Specifies the name of a boundary.

Offset
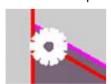
Specifies an offset distance from the boundary. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

NOTE Invalid text displays with a gray background in the **Name** box or the **Offset** box.

Apply

Displays a preview of the boundaries in the graphic view.

NOTE Gaps between boundaries display with a glyph.

The glyph uses the **Handle** color defined in **Tools** > **Options**. For more information, see *Colors Tab (Options Dialog Box)* in the *Common User's Guide.*

OK

Closes the dialog box and saves the boundary changes.

NOTE To cancel boundary changes, you must click **OK** and then exit the parent command without clicking **Finish**.

Selecting Objects

All objects in the Structural Detailing task have properties that you can edit. Using the **Select**
command on the vertical toolbar, you select the object to edit or work with.

An important part of the **Select** command is the **Locate Filter** box that appears on the ribbon. The **Locate Filter** box contains the available, predefined filters for the **Select** command. When you choose a filter in the **Locate Filter** box, the software allows you to select only the filtered objects in a graphic view and in the **Workspace Explorer**. For example, if you select **Seams**, you can select only seams in a graphic view or in the **Workspace Explorer**.

The Structural Detailing task includes these filters:

Assembly Connections/Free End Cuts

Limits your selection in a graphic view or in the **Workspace Explorer** to assembly connections or free end cuts.

Beam Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to beam parts.

Construction Graphics

Limits the selection of items to construction graphics.

Features

Limits your selection in a graphic view or in the **Workspace Explorer** to manually-placed and automatically-generated features. Manually placed features include corner features, edge features, and sketched features. Automatically placed features include slots, collars and clips, and chamfers.

Physical Connections

Limits your selection in a graphic view or in the **Workspace Explorer** to physical connections.

Physical Connections Split Point

Limits your selection in a graphic view or in the **Workspace Explorer** to physical connection split points.

Plate Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to plate parts.

Profile Edge Reinforcement Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to profile edge reinforcement parts.

Seams

Limits your selection in a graphic view or in the **Workspace Explorer** to straking seams.

Stiffener Parts

Limits your selection in a graphic view or in the **Workspace Explorer** to stiffener parts.

Struct Detailing Entities

Allows you to select plate parts, stiffeners parts, beam parts, profile edge reinforcement parts, seams, features, assembly connections, and physical connections in a graphic view and in the **Workspace Explorer**. Objects placed using other tasks, such as pieces of equipment, cannot be selected using this filter.

All

Allows you to select any object, even objects created in another task.

Inside Fence

Selects all objects entirely inside the fence.

Inside/Overlapping Fence

Selects all objects entirely inside the fence and those objects outside but touching the fence at some point.

Split Notification

When a seam is added, deleted, or modified, the plate or profile systems and parts split by the seam are affected. The software notifies different tasks to transfer attributes and objects to the new systems and parts. The Molded Forms, Structural Detailing, Planning, Hole Management, and Structural Manufacturing tasks are affected by split notification.

The software uses split notification when:

- A design or planning seam is added and **Execute Split** is run in the Molded Forms task.

- A design or planning seam that has already been split is deleted or modified in the Molded Forms task.

- An intersection seam is added and split when **Execute Split** is run in the Molded Forms task.

- A profile or plate system creating an intersection seam is deleted or modified in the Molded Forms task.

- A planning seam is added when **Manage Block Intersections** is run and an intersecting part is set to **Split** or **Offset** in the Planning task.

- A planning seam is deleted when **Manage Block Intersections** is run and an intersecting part is set to **Intersected** or **Assigned** in the Planning task.

- A planning seam is modified when the cutting plane of a block is modified in the Planning task.

Identifying Seams in Workspace Explorer

You can identify the type of seam in the **Workspace Explorer** by the icon.

- Design seams

- Planning seams

- Straking seams

- Intersection seams
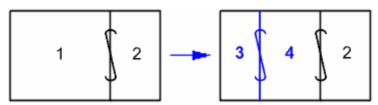
What do you want to do?

- *Add a seam* (on page 51)
- *Delete a seam* (on page 52)
- *Modify a seam when leaf systems are not added or removed* (on page 53)

Add a seam
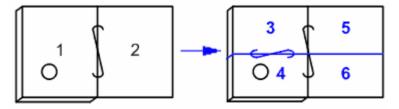
If you split a leaf system and its part with a new seam, the software creates new leaf systems and parts for each split, and deletes the original leaf system.

Molded Forms properties and material information are transferred to the new systems. The default naming rule is used to name the new leaf systems. Logical connections that cross the split are replaced by two new connections with the same properties.
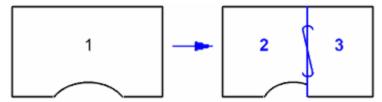
If the part is detailed, Structural Detailing properties and features are transferred to the new parts. The software creates new assembly and physical connections using properties from the previous connections.

⚠ **CAUTION** Straking seams, the leaf parts created by straking seams, and free edge treatments are not transferred and are deleted.
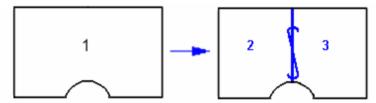
If a seam intersects an existing Structural Detailing feature, the feature is transferred to one of the parts.

Hole cuts and structural hole fittings in the Hole Management task are placed on the **To Do List**. Hole traces and outfitting catalog hole fittings are transferred to the new parts.

If a split passes through an edge feature definition point, the software creates two edge features. If the split does not pass through the definition point, then only the part containing the definition point contains an edge feature.

Block and assembly assignments in the Planning task are transferred. New parts are in the same assembly or block as the original part.

Manufacturing parts are updated using the **Manufacturing Service Manager**.

Connections to ladders, stairs, handrails, members, hangers, and equipment are transferred to the new part.
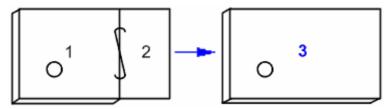
Delete a seam

If you delete or rebound a seam, the software deletes leaf systems and parts and creates new leaf system and parts.

Molded Forms properties and material information are transferred from the first system (as defined by the software) to the new system. The default naming rule is used to name the new leaf system. Logical connections that crossed the split are replaced by a new connection with the same properties as the first system.

Structural Detailing properties and features are transferred to the new part. If one part is detailed and the other is not, the new part is detailed.

⚠ **CAUTION** Straking seams, the leaf parts created by straking seams, and free edge treatments are not transferred and are deleted.

Hole cuts and structural hole fittings in the Hole Management task are placed on the **To Do List.** Hole traces and outfitting catalog hole fittings are transferred to the new parts.
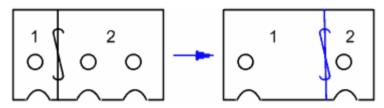
Block and assembly assignments in the Planning task are transferred. If the original parts were in the same assembly or block, then the new part is placed in that assembly or block. If the original parts were in different assemblies, then the new part is placed in the assembly last used by split notification.

Manufacturing parts are updated using the **Manufacturing Service Manager**.

Connections to ladders, stairs, handrails, members, hangers, and equipment are transferred to the new part.

Modify a seam when leaf systems are not added or removed

If you move a seam and the existing systems and parts are modified instead of replaced, then the software does not use split notification. **Structural Detailing** features are not transferred from one part to another, and are placed on the **To Do List**. If a feature is on one part and a seam is later modified so that it crosses the feature, the feature is still on one part. Split notification is not used when a seam is modified.

🗐 NOTES

- Split notification works with design seams from Molded Forms as well as with planning seams.

- If there are multiple features on one sketch, the software creates two sketches after the split; one for each part. Constraints associated with a sketched feature only go with one sketch after the split. You must manually add the constraints to the other sketch.

S ECTION 3

Execute Detailing Command

The **Execute Detailing** command creates detail parts from the root part of a plate, profile, or member system. The command requires you to select one or more blocks, assemblies, systems, sub-systems, or root parts. You can select built-up members for detailing, but you cannot select rolled members. You can select the objects from the model, or from the **Workspace Explorer**.

Objects can be selected for detailing both before and after launching the **Execute Detailing** command. Only valid items are added to the list and are highlighted in yellow, and invalid commands are filtered out.

After you run the **Execute Detailing** command, the icon for the object changes in the **Workspace Explorer**. For more information, see *Icons in the Workspace Explorer* (on page 19).

When parts are created in the Molded Forms task, those parts are trimmed to the molded surfaces of the part's boundaries. Parts in this initial state are referred to as *light* parts. The **Execute Detailing** command transforms the light parts to production definition and accuracy by considering the thickness, orientation, and connection information of the bounding objects. Parts in this advanced state are referred to as *detailed* parts. In addition to the thickness based trimming, other objects, such as assembly and physical connections, slots, collars, end cuts, and chamfers are automatically created. These additional objects are either incorporated in the detail part, or become related to the detailed part, as appropriate.

The software trims a detail part based on its bounding objects. After a bounding object has been detailed, the resulting detail part boundary is trimmed to the face of the bounding detail part, which means the boundary is adjusted by the thickness of the bounding detail part. If the bounding object has not been detailed (that is, the **Execute Detailing** command has not been run for its root part), then the resulting detail part boundary is trimmed to the bounding parent plate system surface. When you run **Execute Detailing** against the bounding root part, the boundary of the bounded detail part is adjusted as required by the molded thickness direction and the thickness of the bounding detail part.

As parts are detailed, additional detailing objects are created as smart occurrences by rules in the reference data and can be individually modified after creation. Assembly and physical connections are created for both parts associated with a connection. Free end cuts are created at the ends of unbounded profiles or members. If you have run **Execute Split** in the Molded Forms task, the software automatically creates slot and collar features where stiffeners penetrate plates, end cuts on profiles, and chamfers at design seams where plate thickness change by a specified amount. The default values of smart occurrence objects cannot be modified with the **Execute Detailing** command, but can be modified after creation by selecting the object and modifying the properties of the object.

If you select a detailed part and click **Delete**, the software removes the detailing from that part.

If you have run **Execute Detailing** for only one of the root parts, then the software does not create the assembly and physical connections or any applicable features. For example, if you run **Execute Detailing** for a root part that is penetrated by a stiffener that has not had **Execute Detailing** run, then the assembly and physical connections for the penetration are not created, and thus the slot and collar or clip features created by the assembly connection are also not created. When you run **Execute Detailing** for the penetrating root part, the assembly

connection at the penetration is created. The software then creates the slot on the penetrated detail part.

The generated detail parts inherit their properties from the parent plate or profile system.

NOTE Straking seams can only be placed on detail parts. Therefore, you must run **Execute Detailing** before you can place straking seams.

Execute Detailing Ribbon

Displays the controls used to generate detail parts.

Select objects for detailing

Select the objects from which to generate detail parts. You can select the objects in a graphic view or in the **Workspace Explorer**.

Finish

Creates the detail parts using the parameters that you have defined.

Plate Systems and Parts Only

Excludes profiles from the parts that the software processes. If you select this option before you select parts, the software prevents you from selecting profile objects. If you select this option before you click **Finish**, the software excludes any profiles that are in your selected objects from processing.

Reject

Clears all selected objects.

Accept

Accepts all selected objects.

Propagate changes

In addition to detailing the selected objects, other parts affected by the detailing are also changed. If many parts are affected, propagation can significantly impact processing time.

Do not propagate changes

Only the selected objects are detailed. Other parts affected by the detailing are not changed, keeping processing time to a minimum.

NOTES
- You can select plate or profile parts by default in a graphic view, or from the **Workspace Explorer**.
- You can select planning blocks by default in a graphic view, or from the **Workspace Explorer**.
- You can select plate or profile systems and sub systems in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all child parts.
- You can select planning blocks and assemblies in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all assigned parts.
- Select the **Plate Systems & Parts Only** option to exclude profiles from the objects that you can select.

- If you select root parts, then only the root parts selected are detailed. For example, selecting a root plate part does not detail the root profile parts on the plate, even if **Plate Systems and Parts Only** is turned off.

Smart Occurrence Ribbon

Displays the controls used to modify a smart occurrence object. The ribbon is available after the smart occurrence object is created automatically by the **Execute Detailing** command.

Properties

Activates the **Properties** dialog box, which you use to view and modify the properties of the smart occurrence object that you are modifying. The initial properties default from the Structural Detailing rules used by the **Execute Detailing** command.

Item

Displays the final result of User Answers on the **Selection** tab of the **Properties** dialog box.

NOTE **Item** does not display for some smart occurrence objects.

Modify Auxiliary Parts Ribbon

Displays the options that you use to add auxiliary parts to the selected assembly connection.

Properties

Activates the **Assembly Connection Properties Dialog Box**, which you use to view and modify the properties of the assembly connection before the assembly connection is committed to the database. For more information, see *Assembly Connection Properties Dialog Box* (on page 289).

Assembly Connection Smart Item

Displays valid smart items for the selected assembly connection.

Add/Delete Auxiliary Parts

Select the parts that can be connected to the selected assembly connection and do not already have an existing assembly connection. Parts that do not have common geometry with the selected assembly connection cannot be selected.

Accept

Adds parts from selected parts to the assembly connection and saves the changes to the database.

Reject

Removes any selected parts from the auxiliary parts list. Only selections made after the last commit are removed.

What do you want to do?

- *Assign part names at placement* (on page 57)
- *Create detailed parts from plate and profile systems* (on page 57)
- *Delete a smart occurrence object* (on page 58)

- *Modify a smart occurrence object* (on page 58)
- *Remove detailed parts from plate and profile systems* (on page 58)
- *Add auxiliary parts to an assembly connection* (on page 287)
- *Edit assembly connection parts* (on page 288)

Assign part names at placement

Every task that creates new parts in the model must assign a part name to each new part.

In the Structural Detailing task, the part name is automatically generated using the default name rule when you place the part. The name consists of the system parent, the category of the part, and a sequencing number.

You can change the part name on the *Main Tab* (on page 412) for each part. When you override the automatically generated name in the **Name** box on the **Main** tab, the text in the **Rule** box changes to **User Defined**.

Create detailed parts from plate and profile systems

1. Click **Execute Detailing** 🥧.

2. Select the parts to detail.

   ♦ **TIPS**

   - You can select plate or profile parts by default in a graphic view, or from the **Workspace Explorer**.

   - You can select planning blocks by default in a graphic view, or from the **Workspace Explorer**.

   - You can select plate or profile systems and sub systems in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all child parts.

   - You can select planning blocks and assemblies in the **Workspace Explorer** or by using **QuickPick** in a graphic view. **Execute Detailing** details all assigned parts.

   - Select the **Plate Systems & Parts Only** option to exclude profiles from the objects that you can select.

   - If you select root parts, then only the root parts selected are detailed. For example, selecting a root plate part does not detail the root profile parts on the plate, even if **Plate Systems and Parts Only** is turned off.

3. Click **Finish**.

Delete a smart occurrence object

1. Click **Select** 🔍 on the vertical toolbar.

2. Select the object to delete.

3. Click **Delete** ✕.

Modify a smart occurrence object

1. Click **Select** 🔍 on the vertical toolbar.

2. Select one of the following in the **Locate Filter** box:

   ▪ **Features** to select slots, collars, web cuts, flange cuts, or chamfers

   ▪ **Assembly Connections / Free End Cuts**

   ▪ **Physical Connections**

3. Select the smart occurrence object to modify.

4. Click **Properties** 📄.

5. Edit properties and parameters for the object as needed.

Remove detailed parts from plate and profile systems

1. Select one or more detailed parts.

   📝 NOTE Detailed parts have red icons in the **Workspace Explorer**. For more information, see *Icons in the Workspace Explorer* (on page 19).

2. Click **Delete** ✕.

   The detailed part is deleted, and replaced by a light part.

Detailing Built Up Connections

The following cases provide examples of the results that you can expect when the software details member to member connections involving built up (designed) members.

📝 NOTE The following abbreviations are used in these descriptions:

▪ M# - Member System

▪ FC# - Frame Connection

▪ DM# - Designed Member

▪ P# - Plate System

▪ AC# Assembly Connection

▪ PC# Physical Connection

▪ P#' - Plate sub System

▪ PP# - Plate Part

- LC# Logical Connection

- LC#' - Sub Logical Connection

Topics

Designed Member to Designed Member - Axis Along

Member2 (DM2) to Member1 (DM1) Axis Along

Object Hierarchy Before Detailing
```
MS1
  -DM1
    -PS11
      -PS11'
        -PP11
    -PS12
      -PS12'
```

```
            -PP12
      -PS13
         -PS13'
            -PP13
   -FC1
   -FC2
MS2
   - DM2
      - PS21
         - PS21'
            - PP21
         - LC 1 (PS21 <-> MS1)
            - LC' 1 (PS21'<-> MS1)
      - PS22
         - PS22'
            - PP22
         - LC 2 (PS22 <-> MS1)
            - LC' 2 (PS22'<-> MS1)
      - PS23
         - PS23'
            - PP23
         - LC 3 (PS23 <-> MS1)
            - LC' 3 (PS23'<-> MS1)
   - FC3
   - FC4
```
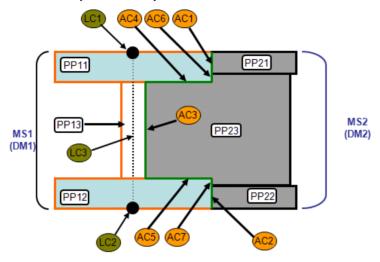
Object Hierarchy After Detailing

```
MS1
  - …
MS2
  - DM2
    - PS21
      - PS21'
        - PP21
      - LC 1
        - LC' 1
          - AC1 (PP21 <-> PP11)
            - PC1
    - PS22
      - PS22'
        - PP22
      - LC 2
        - LC' 2
          - AC2 (PP22 <-> PP12)
            - PC2
    - PS23
      - PS23'
        - PP23
      - LC 3
        - LC' 3
          - AC3 (PP23 <-> PP13)
            - PC3
          - AC4 (PP23 <-> PP11)
            - PC4
          - AC5 (PP23 <-> PP12)
            - PC5
  - FC3
  - FC4
```

Flange Thickness of Supported Member is Less Than Flange Thickness of Supporting Member
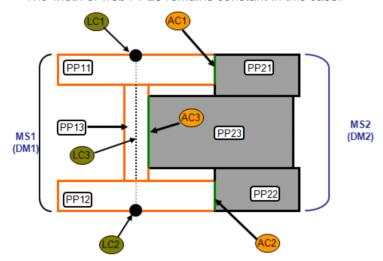
📄 **NOTE** Object hierarchy is not shown.

Flange Thickness of Supported Member is Greater than Flange Thickness of Supporting Member
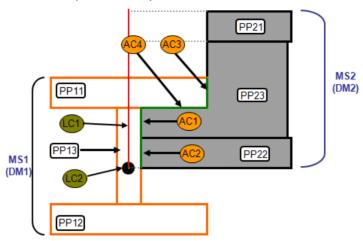
📄 **NOTES**

- Object hierarchy is not shown.
- The width of web PP23 remains constant in this case.

Differing Control Points

⊟ NOTE Object hierarchy is not shown.
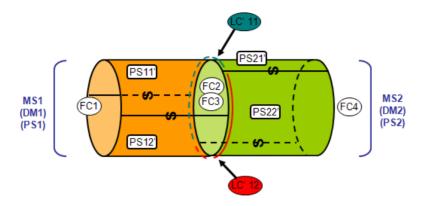
Tube End to Tube End - Co-Linear Axis

Member1(DM1)[Tube] then continue to Member2 (DM2) [Tube] OD Different, ID Same

Object Hierarchy Before Detailing
```
MS1
  – DM1
    – PS1
      – PS11'
        – PP11
      – PS12'
        – PP12
      – Seam1
      – Seam2
  –FC1
  –FC2
MS2
  – DM2
    – PS2
      – PS21'
        – PP21
      – PS22'
        – PP22
      – Seam1
      – Seam2
      – LC 1 (PS2 <-> MS1)
        – LC' 11 (PS21' <-> MS1)
        – LC' 12 (PS22' <-> MS1)
    – FC3
    – FC4
```

Object Hierarchy After Detailing

```
MS1
  – DM1
    – PS1
      – PS11'
        – PP11
      – PS12'
        – PP12
      – Seam1
      – Seam2
  –FC1
  –FC2
MS2
  – DM2
    – PS2
      – PS21'
        – PP21
      – PS22'
        – PP22
      – Seam1
      – Seam2
      – LC 1 (PS2 <-> MS1)
        – LC' 11 (PS21' <-> MS1)
          – AC111 (PP21 <-> PP11)
            – PC111
          – AC112 (PP21 <-> PP12)
            – PC112
        – LC' 12 (PS22' <-> MS1)
          – AC121 (PP22 <-> PP11)
            – PC121
          – AC122 (PP22 <-> PP12)
            – PC122
  – FC3
  – FC4
```
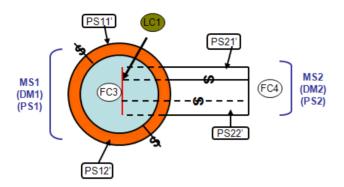
Tube to Tube - Axis Along

Member1(DM1)[Tube] connected to Member2 (DM2) [Tube/Cone] Axis Along Member2 into the side of Member1

📑 NOTE Closed surfaces, such as cones and tubes, must have at least two longitudinal design splits for the software to detail them.

Object Hierarchy Before Detailing

```
MS1
  – DM1
    – PS1
      – PS11'
        – PP11
      – PS12'
        – PP12
      – Seam1
      – Seam2
  –FC1
  –FC2
MS2
  – DM2
    – PS2
      – PS21'
        – PP21
      – PS22'
        – PP22
      – LC 1 (PS2 <-> MS1)
        – LC' 11 (PS21'<-> MS1)
        – LC' 12 (PS22'<-> MS1)
  – FC3
  – FC4
```

Object Hierarchy After Detailing

```
MS1
  – DM1
    – PS1
      – PS11'
        – PP11
      – PS12'
        – PP12
      – Seam1
      – Seam2
  -FC1
  -FC2
MS2
  – DM2
    – PS2
      – PS21'
        – PP21
      – PS22'
        – PP22
      – LC 1 (PS2 <-> MS1)
        – LC' 11 (PS21'<-> MS1)
          – AC1
            – PC1
        – LC' 12 (PS22'<-> MS1)
          – AC2
            – PC2
  – FC3
  – FC4
```

See Also

Detailing Built Up Connections (on page 58)
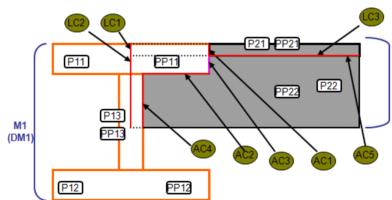
Designed Member to Designed Member

Member2(DM2) to Member1 (DM1) Axis Along (Flush on top flange, DM2 smaller depth and smaller flange thickness)

⊟ NOTE The frame connection is the same as in .*Designed Member to Designed Member - Axis Along* (on page 59)

See Also

Detailing Built Up Connections (on page 58)

Designed Member to Designed Member - Axis Along
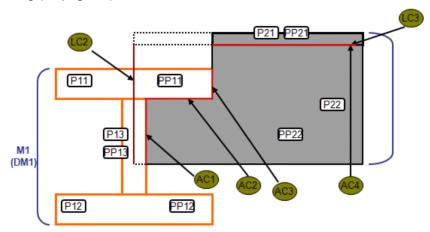
Member (DM2) to Member (DM1) Axis Along (M2 offset above M1)

⯊ NOTE The frame connection is the same as *Designed Member to Designed Member - Axis Along* (on page 59).

See Also
Detailing Built Up Connections (on page 58)

Designed Member to Designed Member - Axis Along
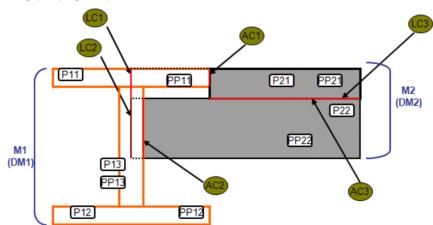
Member(DM2) to Member (DM1) Axis Along (Flush on top flange, M2 flange thickness > M1 flange thickness)
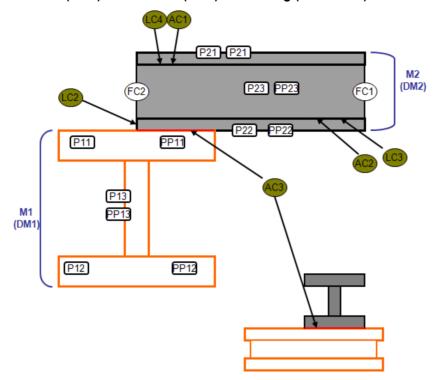
⯊ NOTE The frame connection is the same as *Designed Member to Designed Member - Axis Along* (on page 59).

See Also
Detailing Built Up Connections (on page 58)

Designed Member to Designed Member - I Seated

Member2 (DM2) to Member1 (DM1) Axis Along ("I-Seated")

See Also
Detailing Built Up Connections (on page 58)
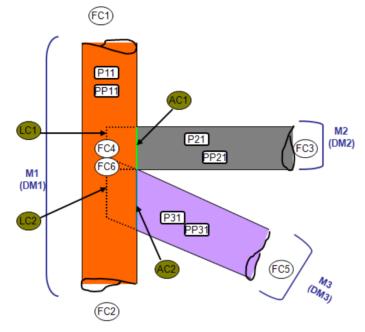
Vertical Brace or Gap

Vertical Brace or Gap 1 or Gap2
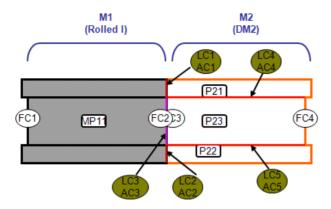
📝 NOTE For Gap2, FC4 and FC6 are Gap2.

See Also
Detailing Built Up Connections (on page 58)
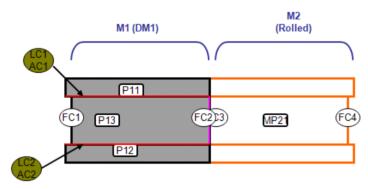
Rolled I to Built Up I

Member 1(Rolled-I) then continue to Member 2 (DM2) [BuiltUp-I]
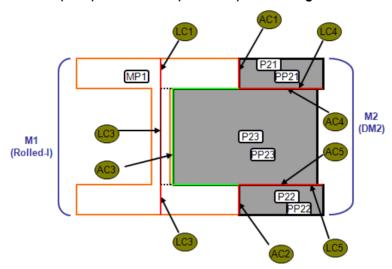
Member 1(DM2) [BuiltUp-I] then continue to Member 2 (Rolled-I)

See Also
Detailing Built Up Connections (on page 58)
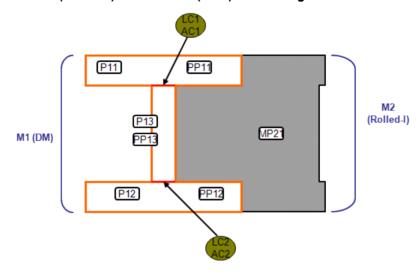
Designed Member to Rolled I

Member2 (DM2) to Member1 (Rolled - I) Axis Along
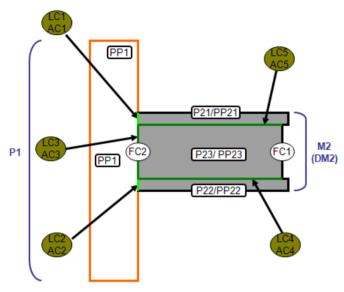
Member2(Rolled-I) to Member1 (DM1) Axis Along

Designed Member to Plate

Member2 (DM2) to Plate (P1) {Surface Connection}

See Also
Detailing Built Up Connections (on page 58)
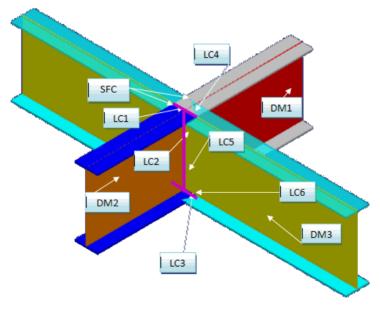
Split Built Up by Built Up - 90° Center
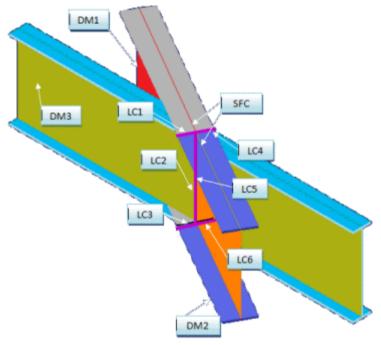
DM1 and DM2 were created by splitting the original design member by DM3. Both members were defined using cardinal point 5.

See Also
Detailing Built Up Connections (on page 58)

Split Built Up by Built Up - Non Perpendicular Center
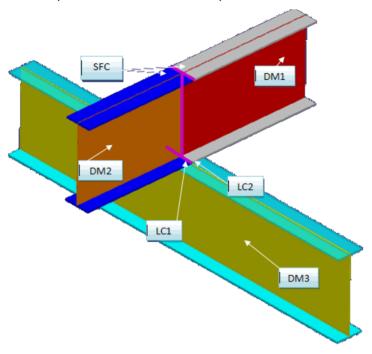
DM1 and DM2 were created by splitting the original design member by DM 3. Both members were defined using cardinal point 5.

See Also
Detailing Built Up Connections (on page 58)

Split Built Up by Built Up - 90⁰ Opposite Connect Points

DM1 and DM2 were created by splitting the original design member by DM3. DM1 and DM2 use cardinal point 2. DM3 uses cardinal point 8.
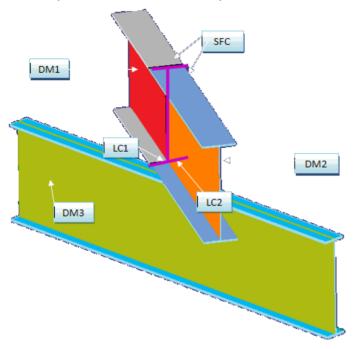
🗐 NOTES

- This configuration is modeled with DM1 and DM2 using cardinal point 2 and DM3 using cardinal point 8. There is not a frame connection between the crossing members before the split. The member axes must intersect for the software to create the split. If there is a gap between the two crossing systems, the software cannot create the logical connections. Structural Detailing requires a connection to detail this case. The lapped property on the structural framing connection is sufficient for this.

- The orientation of the split is determined by the plane through the intersection point of the two member axes. This plane is normal to the split member axis.

- Structural Detailing does not create a connection between DM1 and DM2.

- If you place manual logical connections between the plate systems for DM1 and DM2, then assembly connections and physical connections are created as children of the manual logical connections.

See Also
Detailing Built Up Connections (on page 58)

Split Built Up by Built Up - Non Perpendicular Opposite Connect Points

DM1 and DM2 were created by splitting the original design member by DM3. DM1 and DM2 use cardinal point 2. DM3 uses cardinal point 8.
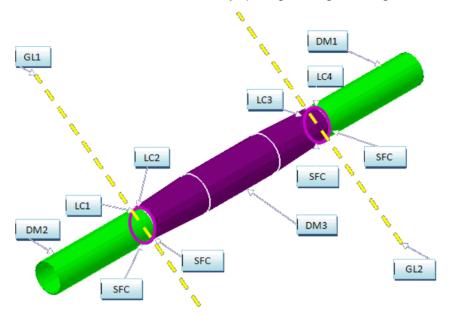
NOTES

- The orientation of the split is not modified by detailing. The trim of the lower flanges for DM1 and DM2 are normal to the direction of the member.

- If manual logical connections are placed between plate systems for DM1 and DM2, then assembly connections and physical connections are created as children of the manual logical connections.

See Also
Detailing Built Up Connections (on page 58)

Split Built Up by Grid - Perpendicular

DM2, DM2, and DM3 were created by splitting the original design member by GL1 and GL2.
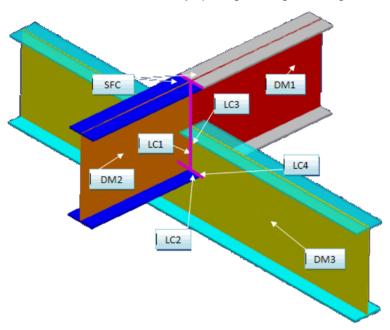
NOTES

- Assembly connections and physical connections are created as children of the logical connections.

- The software checks to ensure than duplicate assembly connections and physical connections are not created. If duplicate logical connections exist, the assembly connections, physical connections, and chamfers are created as children of the first logical connection processed.

See Also
Detailing Built Up Connections (on page 58)
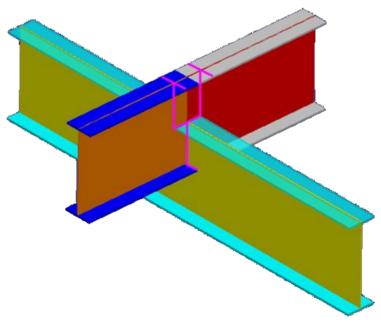
Split Built Up by Built Up - 90º Overlapping Connect Points

DM1 and DM2 were created by splitting the original design member by DM3.
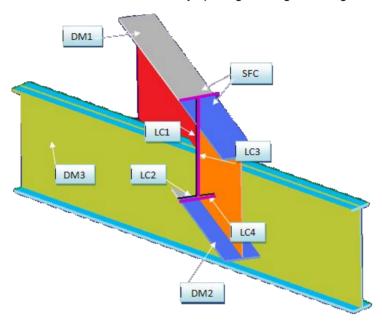
The following figure indicates the trim lines after detailing:

See Also
Detailing Built Up Connections (on page 58)
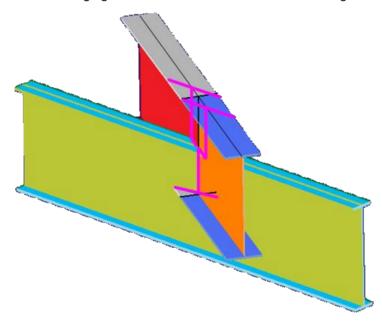
Split Built Up by Built Up - Non Perpendicular Partial Penetration

DM1 and DM2 were created by splitting the original design member by DM3.
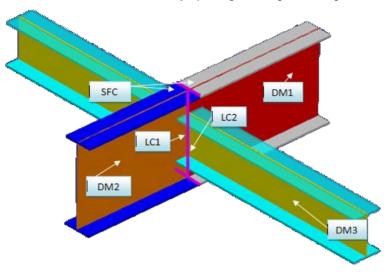
The following figure indicates the trim lines after detailing:

See Also
Detailing Built Up Connections (on page 58)

Split Built Up by Built Up - 90º Penetration

DM1 and DM2 were created by splitting the original design member by DM3.

The following figure indicates the trim lines after detailing:

See Also
Detailing Built Up Connections (on page 58)
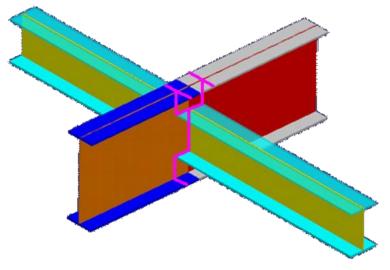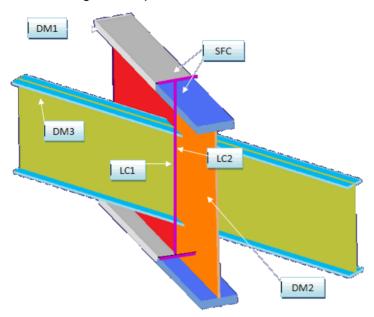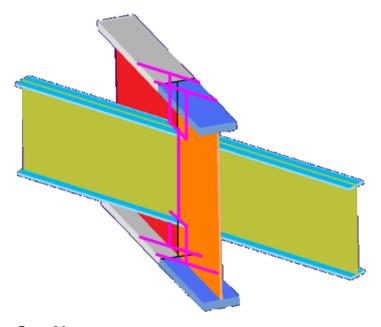
Split Built Up by Built Up - Non Perpendicular Penetration

DM1 and DM2 were created by splitting the original design member by DM3. Both members are defined using cardinal point 5.

The following figure indicates the trim lines after detailing:

See Also

Detailing Built Up Connections (on page 58)
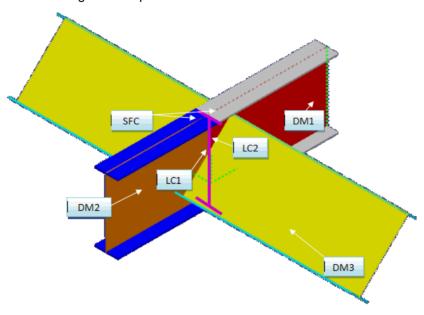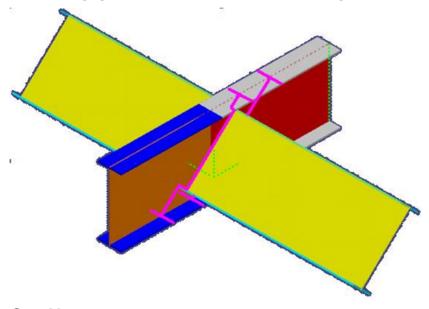
Split Built Up by Built Up - 90⁰ Rotation of Member

DM1 and DM2 were created by splitting the original design member by DM3. Both members are defined using cardinal point 5.

The following figure indicates the trim lines after detailing:

See Also

Detailing Built Up Connections (on page 58)
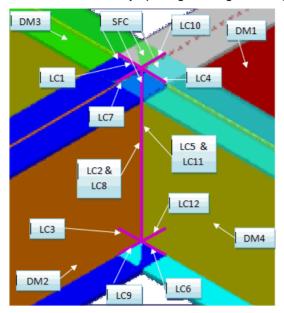
Split Built Up by Built Up - 90⁰ Split Both
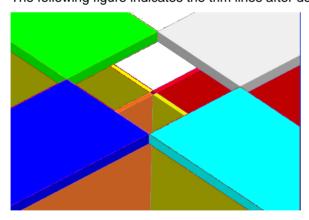
DM1 and DM2 were created by splitting the original design member by DM3 and DM4. DM3 and DM4 were created by splitting the original design member by DM1 and DM2.

📄 NOTES

- Structural Detailing creates only the split frame connections. This is an ambiguous case that you need to modify later in the design cycle with additional objects such as column or side plates, so logical connections are not created.

- All assembly and physical connections are placed on the **To Do List** because there is no physical contact after trimming.

The following figure indicates the trim lines after detailing:
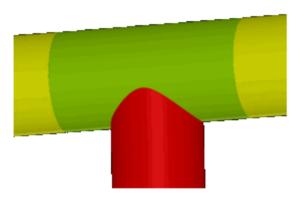
See Also
Detailing Built Up Connections (on page 58)

Can - Horizontal Beam

The beam is horizontal and continuous before the can is placed.

📝 NOTES

- The beam can be a built-up member.

- The beam can be a standard member.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.
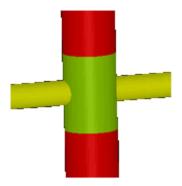
See Also
Detailing Built Up Connctions (on page 58)

Can - Vertical Column

📝 NOTES

- The column is continuous before the can is placed.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.
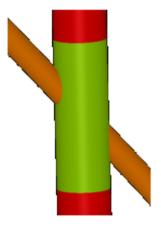
See Also
Detailing Built Up Connections (on page 58)

Can - Continuous Leg

NOTES

- The leg is continuous.

- The column is doubly sloping (sloping X-Z and Y-Z plane).

- The column is continuous before the can is placed.

- The software does not support an incoming continuous leg. The leg must be bound to the column that has the can inserted.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

See Also
Detailing Built Up Connections (on page 58)
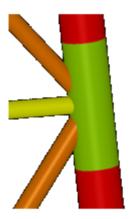
Can - Multiple Braces

NOTES

- The column is doubly sloping (sloping X-Z and Y-Z plane).
- The column is continuous before the can is placed.
- The beam is a built-up member.
- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.
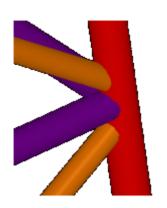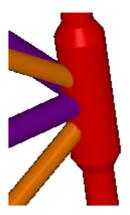
See Also
Detailing Built Up Connections (on page 58)

Can - Diameter Change

NOTES

- The column is doubly sloping (sloping X-Z and Y-Z plane).
- The column is continuous before the can is placed.
- The can is placed with an outer diameter that is larger than either leg.
- The beam is a built-up member.
- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).
- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

See Also
Detailing Built Up Connections (on page 58)
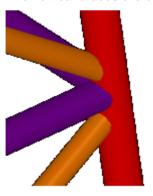
Can - Single Axis Gap

📝 NOTES
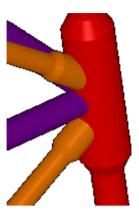
- The column is doubly sloping (sloping X-Z and Y-Z plane).

- The column is continuous before the can is placed.

- Rule-based gaps are applied (AxisGap - Single) to the vertical braces shown in orange before the can is placed.

- The vertical braces and beams are all coplanar.

- The beam is a built-up member.

- The can is placed with an outer diameter larger than either leg.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

Two Vertical Braces with Gap Applied
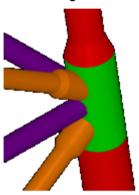
The vertical braces are coplanar. They are smaller than the can and are built-up tubes.

Stub End Cans Added to Vertical Braces

Bottom Leg Matches the Outer Diameter of the Can

See Also
Detailing Built Up Connections (on page 58)
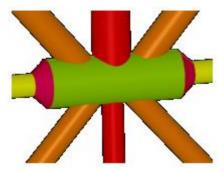
Can - Multiple Braces

NOTES

- The beam is continuous before the can is placed. Only the columns (shown in red) exist on either side.

- An oversized can is applied to the beam (shown in yellow).

- Braces are added to the can with rule based gaps (AxisGap - Single).

- The vertical braces, columns, and beams are all coplanar.

- The cone is set to have a matching ID with the can.

- Members are built-up members.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.
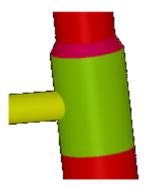
See Also
Detailing Built Up Connections (on page 58)

Can - Cone Transition

🗒 NOTES

- The bottom leg and top leg are of different sizes.

- The can matches the outer diameter of the bottom leg.

- The cone matches the outer diameter of the top leg.

- The column is continuous before the can is placed.

- The beam is a built-up member

- The can is inserted first, then the leg sizes are changed.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

See Also
Detailing Built Up Connections (on page 58)
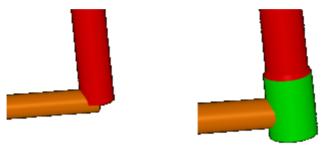
Can - Beam and Column End Point

NOTES

- The beam and the column share the same end point.

- The can has a larger outer diameter than the outer diameter of the leg.

- The beam is a built-up member.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

See Also
Detailing Built Up Connections (on page 58)

Can - Beam and Column Intersection

NOTES

- The can matches the outer diameter and ID of the column.

- The beam is horizontal and split before the can is placed.

- For more information on the chamfer properties, see *Chamfer Properties Dialog Box* (on page 432).

- Inserting cans may result in two root logical connections with duplicate geometry. However, Assembly Connections and Physical Connections are only created for one of the Logical Connections.

See Also
Detailing Built Up Connections (on page 58)

S E C T I O N 4

Place Plate Parts

Creates standalone detailed plate parts. These plate parts do not have parent systems. They are generally used for small plates that are of secondary structural importance, but are not limited to just those uses.

You complete three steps when you are creating a plate system. First, use the **Properties** command for defining properties and defaults for the plate part. Next, define the plane of the plate. Then, define topological boundaries. You can define the topological boundaries by selecting objects in a graphic view, selecting objects from the **Workspace Explorer**, or selecting objects from a list of potential bounding objects.

Place Plate Parts Ribbon

Displays the options that you use to place the standalone detailed planar plate part in the model.

Plate Part Properties

Activates the **Place Plate Part Properties** dialog box, which you use to view and modify the properties of the plate part that you are about to place in the model. The initial plate system properties default from the selected structural specification. For more information, see *Place Plate Parts Properties Dialog Box* (on page 98).

Define Plane

Select the reference plane of the plate part. You can select multiple planes to place multiple plate parts at the same time. The software uses the same boundaries for each plane and places the plate parts. After placement, the plate parts are unrelated to each other and can be edited as individual entities unless boundaries are defined as boundary curves. If so, then a single curve is related to all of the plate parts.

Define Boundaries

Define the outside boundaries of the plate part. You can define the boundaries by selecting objects in the model or by sketching boundary curves. If you select objects in the model, and those objects are moved, the software automatically resizes the plate part to maintain the boundary relationship.

Define Thickness Direction

Specifies the side of the plane on which to place the thickness of the plate.

Finish

Places the plate part using the defined parameters.

Define Plane Controls

By Plane or Offset Plane

Places the plate system a specified distance from the reference plane. If you choose this option, you must define the offset distance. An offset distance of zero defines a coincident plane.

Plane by Point and Vector

Specifies the reference plane by defining a vector perpendicular to the reference plane. A third point defines the reference plane position along the vector.

Plane by Three Points

Specifies the reference plane using three points that you specify in the model.

Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves. The **Lock** option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance for the plate part from the selected reference plane. You can specify the offset dynamically in graphics or by typing the distance. The **Offset** option is only available when you use the **Offset from a Plane** option.

Angle

Specify the angle at which to place the plate part relative to the reference plane. You have to define the axis of rotation using two points before you can define the angle. The **Angle** option is only available when you use the **Angle from plane** option.

Step

Specifies the angle step. The step is incremented by this value when the cursor is moved in the graphic view. The **Step** option is only available when you use the **Angle from plane** option.

Select Vector

Select or define a vector normal to the needed plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point

Specify the point along the vector at which to place the plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point 1

Specify the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 2

Specify the location of the second of three points that defines the plane. This option is only

available when you are using the **Plane by Three Points** option.

Define Point 3

Specify the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Boundary Controls

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

TIP Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Select this option to pick free edges on plates to define the boundary. This option is available if you selected a plate in the **Pick Boundaries** step.

Solve Ambiguity

If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to clarify the needed boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 35).

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 45).

Boundary Curve

Select this option to draw the boundaries. This option is only available when you are defining the boundaries.

Auto

Indicates whether Smart 3D automatically adds all related objects to the select set.

Reject

Clears all selected planes or boundaries.

Accept

Accepts all selected planes or boundaries.

Parent System

Select the parent system for the plate part that you are placing. Systems are defined using

the Systems and Specifications task.

Material

Select the material for the plate part.

Grade

Select the material grade for the plate part.

Thickness

Select the material thickness for the plate part.

What do you want to do?

- *Transfer ownership of objects in the model* (on page 95)
- *Place a standalone planar plate part* (on page 95)
- *Modify a standalone planar plate part* (on page 96)
- *Delete a standalone planar plate part* (on page 96)

Transfer ownership of objects in the model

1. Select one or more objects in the model.
2. Click **Edit > Properties**.
3. On the **Configuration** tab, click **Transfer**.
4. On the **Transfer Ownership** dialog box, specify the new location and permission group.
5. Click **OK**, and click **Yes** on the confirmation message.

NOTES

- You must have physical write permission to the permission group where the objects currently reside.
- You cannot undo the transfer of ownership.
- The other location receives the change when the database replication process for the objects takes place.

Place a standalone planar plate part

1. Click **Place Plate Part** on the vertical toolbar.
2. Specify the plane for the plate part using one of the following methods:

   Define plane using angle from plane (on page 29)

   Define a coincident plane (on page 29)

   Define an offset plane (on page 30)

   Define a plane using three points (on page 31)

Define a plane using a point and a normal vector (on page 32)

3. Click **Accept** ✅.

4. Define the plate system boundaries using one or more of the following methods:

   Pick boundaries (on page 34)

   Define a boundary curve (on page 35)

5. Click **Accept** ✅.

6. Set the remaining plate system properties.

7. Click **Finish**.

📝 **NOTE** If you define an ambiguous boundary, you must provide more information by selecting bounded areas that together compose the total bounded area. For more information, see *Boundary Methods* (on page 33).

Modify a standalone planar plate part

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Plate Parts** in the **Locate Filter** box.

3. Select the planar plate part to modify.

4. Using the ribbon controls, edit the planar plate part as needed.

📝 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a standalone planar plate part

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Plate Parts** in the **Locate Filter** box.

3. Select the planar plate part to delete.

4. Click **Delete** ✖.

Transfer Ownership Dialog Box

Allows you to specify a new location and permission group for the selected model objects.

Current location

Displays the name of the location that the current permission group is associated with. All of the objects in the select set must belong to the same location.

Current permission group

Displays the name of the permission group that the selected objects are currently associated with. If all of the objects in the select set do belong to the same permission

group, this box appears blank.

New location

Specifies the name of the location to which to assign the objects. In a global workshare configuration, this box lists all the locations in which you have write access to one or more permission groups. The selection in this box filters the entries in the **New permission group** box.

New permission group

Specifies the new permission group to which to assign the selected objects. If you have specified a value in the **New location** box, this list displays all permission groups that you have write access to in the selected location. If you have not specified a value in the **New location** box, this list includes all permission groups that you have write access to in all locations except the current location. This box is blank if you do not have write access to any permission groups at any locations other than the current one.
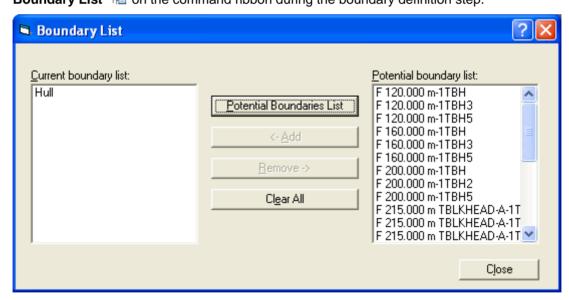
🗐 **NOTE** It is strongly recommended that administrators follow naming convention rules that include the location as a prefix in the permission group name.

See Also
Relationship Tab (on page 400)

Boundary List Dialog Box

The **Boundary List** dialog box displays all the boundaries that you have currently selected and all the potential boundaries that you could select. You activate this dialog box by clicking

Boundary List 🗒 on the command ribbon during the boundary definition step.

Current boundary list

Lists all the selected boundaries. The object that you are placing has these boundaries.

Potential Boundaries List

Opens the **Potential boundaries list**.

Potential boundary list

Lists all boundaries objects that you can select that meet the following criteria:

- Boundaries removed from the **Current boundary list** using the **Remove** button.

- Objects associated with other current boundaries, such as a child plate system of a current plate system boundary.

Add

Adds the selected boundaries in the **Potential boundaries list** to the **Current boundary list**.

Remove

Removes the selected boundaries from the **Current boundary list** and returns them to the **Potential boundaries list**.

Clear All

Removes all boundaries from the **Current boundary list**.

See Also
Use the Boundary List Dialog Box (on page 34)

Place Plate Parts Properties Dialog Box

Specifies the properties for the plate part system that you are editing.

See Also
Main Tab (Place Plate Parts Properties Dialog Box) (on page 98)
Material Tab (on page 100)
Molded Conventions Tab (on page 100)
Transfer Ownership of Objects in the Model (on page 95)

Main Tab (Place Plate Parts Properties Dialog Box)

Specifies the general properties of the standalone planar plate part.

Name

Specifies the name of the plate part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the plate part. Select **User Defined** to type a name in the **Name** box.

Type

Select the type of plate part that you are placing. If you do not specify a type, the software automatically determines the type based on the plate part orientation. Plate parts that are mostly horizontal (XY plane) are assigned to **Deck**. Plate parts that are mostly transverse (YZ plane) are assigned to **Transverse Bulkhead**. Plate parts that are mostly longitudinal (XZ plane) are assigned to **Longitudinal Bulkhead**. This option is also used by the naming rule to name the plate part.

Naming Category

Select a category for the plate part. Categories specify the role of the plate part in the marine structure.

Parent System

Specifies the parent plate system for the plate part. You can define a different parent system in the Systems and Specifications task. When the plate part is created, the property values for the parent plate system are used for the initial plate part property values. When a parent property value changes, the corresponding plate part value is not updated.

Surface Geometry Type

Displays **Planar Plate**, similar to the Molded Forms command used to create a planar plate system.

Specification

Select the structural specification for the plate part.

Description

Type a description for the plate part.

Tightness

Select the level of tightness as it applies to the entire plate part.

Board Management

Symmetry

Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Manually Override

Turn on this option to stop the Board Management Service from considering this part.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

See Also
Place Plate Parts Properties Dialog Box (on page 98)

Material Tab

Specifies the material properties for the plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

See Also

Place Plate Parts Properties Dialog Box (on page 98)
Plate Edge Reinforcement Part Properties Dialog Box (on page 441)

Molded Conventions Tab

Plate System Type

Displays the plate system type specified on the **Main Tab**. The delivered reference data provides different orientation settings for each type. Select **Deck**, **Transverse Bulkhead**, **Longitudinal Bulkhead**, **Hull**, **Longitudinal Tube**, **Transverse Tube**, **Vertical Tube**, **Tube Plate**, **Web Plate**, **Flange Plate**, or **General Plate**.

Fore Aft Reference

Displays the longitudinal location at which the molded conventions change, usually midship. This value is set in the structural specification and cannot change.

Port Starboard Reference

Displays the transverse location at which the molded conventions change, usually the longitudinal centerline. This value is set in the structural specification and cannot change.

Deck Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Above - Positions thickness above the deck. This is the default value.

Below - Positions thickness below the deck.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Up - Orients profiles above the deck.

Down - Orients profiles below the deck. This is the default value.

Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for profiles on the port and starboard sides of the port/starboard reference. The reference is usually the longitudinal centerline.

Inboard - Orients profile flanges towards the port/starboard reference. This is the default value.

Outboard - Orients profile flanges away from the port/starboard reference.

Port - Orients profile flanges to the port side.

Starboard - Orients profile flanges to the starboard side.

Longitudinal Profiles at Port Stbd Reference

Defines the orientation of the flanges for profiles on the port/starboard. The reference is usually the longitudinal centerline.

Port - Orients profile flanges to the port side. This is the default value.

Starboard - Orients profile flanges to the starboard side.

Transverse Profiles Forward of Fore Aft Reference
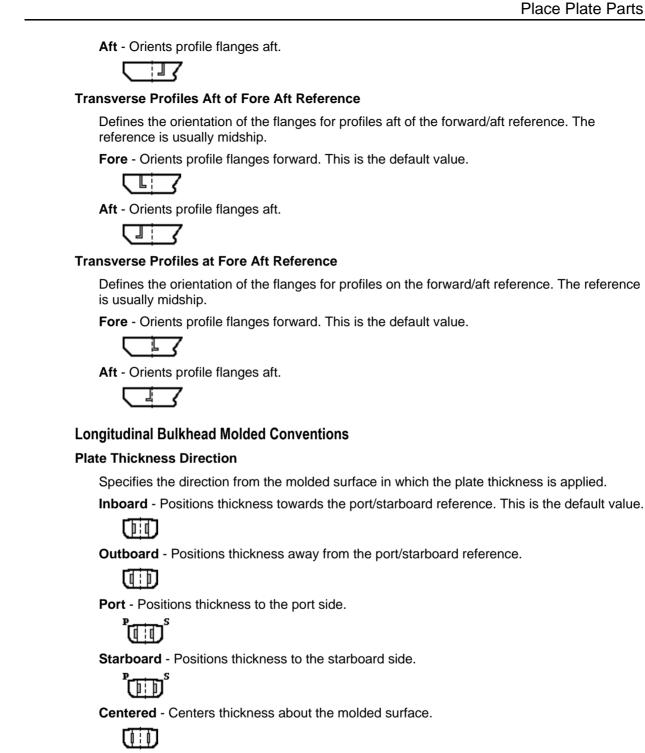
Defines the orientation of the flanges for profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.

Aft - Orients profile flanges aft.

Transverse Profiles Aft of Fore Aft Reference

Defines the orientation of the flanges for profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.

Aft - Orients profile flanges aft.

Transverse Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward. This is the default value.

Aft - Orients profile flanges aft.

Longitudinal Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness towards the port/starboard reference. This is the default value.

Outboard - Positions thickness away from the port/starboard reference.

Port - Positions thickness to the port side.

Starboard - Positions thickness to the starboard side.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference.

Outboard - Orients profiles away from the port/starboard reference. This is the default value.

Port - Orients profiles to the port side.

Starboard - Orients profiles to the starboard side.

Profile Orientation at Port Stbd Reference

Defines the primary orientation for profiles at the port and starboard reference.

Port - Orients profile flanges to the port side.

Starboard - Orients profile flanges to the starboard side.

Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Up - Orients profile flanges upward.

Down - Orients profiles downward. This is the default value.

Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.

Aft - Orients profile flanges aft. This is the default value.

Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference. The reference is usually midship.

Fore - Orients profile flanges forward.

Aft - Orients profile flanges aft. This is the default value.

Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for profiles on the forward/aft reference, usually midship.

Fore - Orients profile flanges forward.

Aft - Orients profile flanges aft. This is the default value.

Transverse Bulkhead Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the forward/aft reference.

Outboard - Positions thickness away from the forward/aft reference.

Forward - Positions thickness to the forward side.

Aft - Positions thickness to the aft side. This is the default value.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Profile Orientation Forward of Fore Aft Reference

Defines the primary orientation for transverse profiles forward of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.

Aft - Orients profiles aft.

Profile Orientation Aft of Fore Aft Reference

Defines the primary orientation for transverse profiles aft of the forward/aft reference.

Fore - Orients profiles forward. This is the default value.

Aft - Orients profiles aft.

Profile Orientation at Fore Aft Reference

Defines the primary orientation for transverse profiles at the forward/aft reference.

Fore - Orients profiles forward. This is the default value.

Aft - Orients profiles aft.

Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Vertical Profiles

Defines the orientation of the flanges for vertical profiles.

Port - Orients profile flanges to the port side.

Starboard - Orients profile flanges to the starboard side.

Inboard - Orients profile flanges toward the port/starboard reference. This is the default

value.

Outboard - Orients profile flanges toward the port/starboard reference

Vertical Profiles on Port Stbd

Defines the orientation of the flanges for vertical profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.

Starboard - Orients profile flanges to the starboard side.

Horizontal Profiles

Defines the orientation of the flanges for horizontal profiles.

Up - Orients profile flanges upward. This is the default value.

Down - Orients profiles downward.

Hull Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

Inboard - Positions thickness toward the port/starboard reference.

Outboard - Positions thickness away from the port/starboard reference. This is the default value.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Inboard - Orients profiles toward the port/starboard reference. This is the default value.

Outboard - Orients profiles away from the port/starboard reference.

Profiles Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Longitudinal Profiles

Defines the orientation of the flanges for longitudinal profiles.

Towards Centerline - Orients profile flanges downward and inboard.

Away From Centerline - Orients profile flanges upward and outboard.

Longitudinal Profile on Port Stbd Reference

Defines the orientation of the flanges for longitudinal profiles on the port/starboard reference.

Port - Orients profile flanges to the port side. This is the default value.

Starboard - Orients profile flanges to the starboard side.

Vertical Profiles Forward of Fore Aft Reference

Defines the orientation of the flanges for vertical profiles forward of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.

Aft - Orients profile flanges aft.

Vertical Profiles Aft of the Fore Aft Reference

Defines the orientation of the flanges for vertical profiles aft of the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.

Aft - Orients profile flanges aft.

Vertical Profiles at Fore Aft Reference

Defines the orientation of the flanges for vertical profiles on the forward/aft reference.

Fore - Orients profile flanges forward. This is the default value.

Aft - Orients profile flanges aft.

Web Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member.

Out - Positions thickness away from the center of the member.

Left - Positions thickness in the negative u-axis direction.

Right - Positions thickness in the positive u-axis direction.

Centered - Centers thickness about the molded surface. This is the default value for designed members.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

Out - Orients profiles away from the center of the member.

In - Orients profiles towards t center of the member.

Left - Orients profiles in the negative u-axis direction.

Right - Orients profiles in the positive u-axis direction. This is the default value for designed members.

Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the member plate web. This is the default value for designed members.

Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the member plate web.

Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the web plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.

Towards End - Orients profile flanges in the direction of the end point of the designed member.

Flange Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the member. This is the default value for designed members.

Out - Positions thickness away from the center of the member.

Up - Positions thickness in the positive v-axis direction.

Down - Positions thickness in the negative v-axis direction.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the member. This is the default value for designed members.

Out - Orients profiles away from the center of the member.

Down - Orients profiles in the negative v-axis direction.

Up - Orients profiles in the positive v-axis direction.

Profiles Secondary Orientation

Axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run parallel to the length of the designed member

In - Orients profile flanges towards the member plate web. This is the default value for designed members.

Out - Orients profile flanges away from the member plate web.

Left - Orients profile flanges in the negative u-axis direction.

Right - Orients profile flanges in the positive u-axis direction.

Non-axial Profiles

Defines the orientation of the flanges for profiles that:

- Are on the flange plate of a designed member
- Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed member. This is the default value for designed members.

Towards End - Orients profile flanges in the direction of the end point of the designed member.

General Plate Molded Conventions

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

With Normal - Positions thickness in the same direction as the molded surface normal.

Opposite Normal - Positions thickness in the direction opposite to the molded surface normal.

Centered - Centers thickness about the molded surface.

Profiles

Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

With Normal - Orients profiles in the same direction as the molded surface normal.

Opposite Normal - Orients profiles in the direction opposite to the molded surface normal.

Secondary Orientation

Specifies the direction of the web thickness for symmetrical cross-sections, or the direction of the web thickness and the flange for unsymmetrical cross-sections.

Left - Orients profile flanges to the left.

Right - Orients profile flanges to the right.

Tube Plate Molded Conventions

Plates

Plate Thickness Direction

Specifies the direction from the molded surface in which the plate thickness is applied.

In - Positions thickness towards the center of the tube member.

Out - Positions thickness away from the center of the tube member. This is the default value for designed tube members.

Centered - Centers thickness about the molded surface.

Offset

Specifies the distance from the molded surface to the first plate face. Type **0** to put the face of the plate on the molded surface. Type a negative value to offset the plate face in the opposite direction of the thickness direction setting. **0** is the default value for designed tube members.

Profiles Primary Orientation

Defines the primary orientation for profiles. Primary orientation defines on which side of the plate system to place the profiles.

In - Orients profiles towards the center of the tube member.

Out - Orients profiles away from the center of the tube member. This is the default value for designed tube members.

Profiles Secondary Orientation

Ring Profiles

Defines the orientation of the flanges for profiles that:

▪ Are on the plate of a designed tube member

▪ Run perpendicular to the length of the designed member

Towards Start - Orients profile flanges in the direction of the start point of the designed tube member. This is the default value for designed members.

Towards End - Orients profile flanges in the direction of the end point of the designed tube member.

Axial Profiles

Defines the orientation of the flanges for profiles that:

▪ Are on the plate of a designed tube member

▪ Run parallel to the length of the designed member

Clockwise - Orients profile flanges in a clockwise direction about the axis of the designed tube member. This is the default value for designed members.
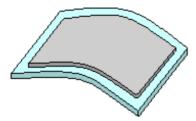
Counter-clockwise - Orients profile flanges in a counter-clockwise direction about the axis of the designed tube member.

SECTION 5

Lapped Plate

📇 Places or modifies a lapped plate on a plate part or standalone plate part. The lapped part derives its surface definition from the part or system to which it is lapped.
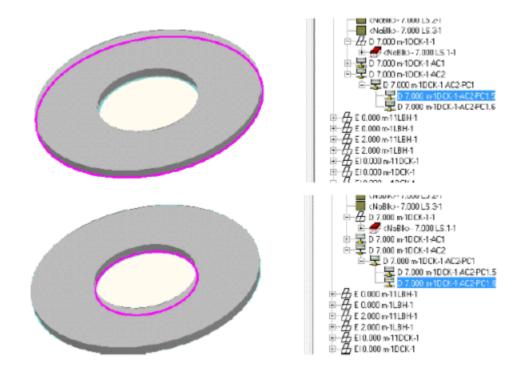
When you create lapped parts, you must select the systems or parts to lap to, define the boundary for the lapped plate, and specify the side of plate. There are several methods available for you to use when you define the boundary for the lapped part. You can graphically pick bounding objects, select the bounding objects from a list of potential bounding objects, or sketch a boundary curve to define the boundaries. You define the side of plate and thickness direction graphically. You can also specify other attributes, such as name rule and material type, for the lapped plate part.

The software creates assembly connections between bounded edges and between the faces of lapped parts. If the plate is lapped to a system, and the lapped plate crosses a seam on the system, two assembly connections are created; one connection to each of the root parts. If the plate is lapped to a system, assembly connections are not created until the root parts have been detailed using **Execute Detailing** 📑.

📋 NOTE You can create only one lapped part at a time.

Lapped Plate Physical Connections

Physical connections are created as children of the lapped plate assembly connections. When you add, delete, or modify a sketched feature on the lapped plate, physical connections are created and updated as needed. A lapped plate with openings has an outer contour and a closed inner contour for each opening. A separate physical connection is created for each contour. These physical connections are children of the parent physical connection of the lapped plate. Different weld parameters can be specified for each physical connection.

Copying a Lapped Plate

If you copy, move, or mirror a lapped plate (including through **Model Data Reuse** operations in the Project Management task), and you do not include a lapped-to part in the copy (by selecting **Edit** > **Paste** > **Delete Optional**), then the software replaces the lapped-to part with a surface body. This prevents the lapped plate from going to the **To Do List**. After the copy, you can replace the surface body with a new plate system or part.

If the original lapped-to part was detailed, **Delete Optional** also deletes the physical connection between the lapped plate and lapped-to plate. For the copied lapped plate, if you replace the lapped-to surface body with a new detailed plate part, then the software creates a new physical connection.

For more information, see *Move*, *Mirror Copy*, and *Paste Dialog Box* in the *Common User's Guide, and Model Data Reuse* in the *Project Management User's Guide*.

Lapped Plate Ribbon

Displays the controls used to place lapped plates.

Properties

Activates the **Lapped Plate Part Properties** dialog box, which you use to view and modify the properties of the lapped plate part that you are about to place in the model. The initial lapped plate properties default from the parent plate system, if there is one, or from the selected structural specification. For more information, see *Lapped Plate Part Properties Dialog Box* (on page 119).

Select Lapped-To Parts

Select the plate system, or standalone plate part to which to lap the new plate. You can select the plate part in a graphic view or in the **Workspace Explorer**.

Boundaries

Define the outside boundaries of the lapped plate. You can define the boundaries by selecting objects in the model or by drawing the boundaries. If you select objects in the model, and those objects are moved, the software automatically resizes the lapped plate to maintain the boundary relationship.

Select Side of Plate

Define the side of the lapped to plate on which to create the lapped plate. You define the side of plate graphically by moving the pointer in the graphic view to change a direction arrow. The lapped part thickness direction is always away from the selected side of the plate part.

Finish

Places the plate part using the defined parameters.

Define Boundary Controls

Pick Boundaries

Select this option to select objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

TIP Pressing the SHIFT key selects the **Pick Edges** option after you have selected a boundary plate. Press SHIFT again to return to the **Pick Boundaries** option.

Pick Edges

Select this option to pick free edges on plates to define the boundary. This option is available if you selected a plate in the **Pick Boundaries** step.

Solve Ambiguity

If the boundaries that you selected can form the object boundary in more than one way, then you have defined an ambiguous boundary. The software prompts you to select multiple bounded areas to clarify the needed boundary. For more information, see *Solve ambiguous solution created by selected boundaries* (on page 35).

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 45).

Boundary Curve

Select this option to draw the boundaries. This option is only available when you are defining the boundaries.

Auto

Indicates whether Smart 3D automatically adds all related objects to the select set.

Define Plane Controls

Offset from Plane

Defines a plane at a specified offset distance from another plane. An offset distance of **0** defines a coincident plane.

Plane by Point and Vector

Defines a plane using a vector normal to the plane being defined. A third point defines the plane position along the vector.

Plane by Three Points

Defines a plane using three points that you identify in the model.

Offset Lock

Locks the **Offset** value, disabling updates of the offset value by moving the pointer in a graphic view. The **Lock** option is only available when you use the **Offset from a Plane** plane method.

Offset

Specifies the offset distance from the selected grid plane or planar model object. You can specify the offset dynamically by moving the mouse in a graphic view or by typing the distance. This option is only available when you use the **Offset from a Plane** plane method. You can type the following formats for offset values:

- Offset distance without units.

  Example: **5**, when default units are millimeters (mm). The software converts the distance to **5 mm**.

- Offset distance with units. The value converts to the default units.

  Example: **5 m**, when default units are millimeters (mm). The software converts the distance to **5000 mm**.

- Coordinate System: Frame plus or minus an offset distance.

  Example (when the model has multiple coordinate systems): **CS_0:F20 + 1.5 m**

  Example (when the model has one coordinate system): **F20 + 1.5 m**

Select Vector

Specifies a vector normal to the required plate system plane. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point

Specifies the point along the vector at which to place the plate system. This option is only available when you use the **Plane by Point and Vector** plane method.

Define Point 1

Specifies the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 2

Specifies the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Define Point 3

Specifies the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** plane method.

Parent System

Select the parent system for the plate part that you are placing. Systems are defined using the Systems and Specifications task.

Material

Select the material for the plate part.

Grade

Select the grade for the plate part.

Thickness

Specifies the thickness of the lapped plate.

Reject

Clears all selected planes or boundaries.

Accept

Accepts all selected planes or boundaries.

What do you want to do?

- *Create a lapped plate part* (on page)
- *Modify a lapped plate part* (on page 119)
- *Delete a lapped plate part* (on page 119)

Create a lapped plate part

1. Click **Lapped Plate** .
2. Select the plate systems or parts to which to lap the new plate.

   TIPS
   - You can select the plate parts in a graphic view or in the **Workspace Explorer**.
   - If you select a curved surface, the lapped part can have only one lapped connection to this system or part. If you select a planar part or system, the lapped part can have lapped connections with multiple coplanar parts or systems.

3. Specify the boundaries for the new plate.
4. Specify the side on which to place the plate.

Modify a lapped plate part

1. Click **Select** 🔖 on the vertical toolbar.

2. Select **Plate Parts** in the **Locate Filter** box.

3. Select the lapped plate system to modify.

   💡 **TIP** You can select the plate parts in a graphic view or in the **Workspace Explorer**.

4. Using the ribbon controls, edit the lapped plate system as needed.

📝 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a lapped plate part

1. Click **Select** 🔖 on the vertical toolbar.

2. Select **Plate Parts** in the **Locate Filter**.

3. Select the lapped plate part to delete.

4. Click **Delete** ✖.

Lapped Plate Part Properties Dialog Box

Specifies the properties for the lapped plate part that you are editing.

See Also

Main Tab (Lapped Plate Part Properties Dialog Box) (on page 119)
Material Tab (Lapped Plate Part Properties Dialog Box) (on page 120)
Configuration Tab (on page 401)

Main Tab (Lapped Plate Part Properties Dialog Box)

Specifies the general properties of the smart occurrence object.

Feature Name

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the **Name** box.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority

codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

See Also
Lapped Plate Part Properties Dialog Box (on page 119)

Material Tab (Lapped Plate Part Properties Dialog Box)

Specifies the material properties for the lapped plate part that you are editing.

Material

Select the material for the plate part.

Grade

Select the material grade for the plate part.

Actual Thickness

Select the material thickness for the plate part.

See Also
Lapped Plate Part Properties Dialog Box (on page 119)

S ECTION 6
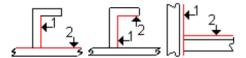
Place Bracket Parts
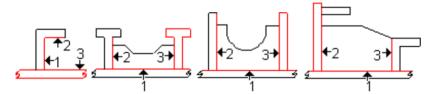
Creates bracket plate parts that reinforce other plate parts. A bracket is always bounded to the structure that it supports. The bracket is a standard structural part and has predefined shapes.

A bracket is classified by its number of supports. The following examples show the general support selection sequence:
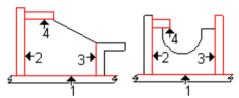
2 Support (2S) Bracket
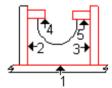
3 Support (3S) Bracket

4 Support (4S) Bracket

5 Support (5S) Bracket

Place Bracket Parts Ribbon

Displays the controls used to place brackets.

Properties

Activates the **Bracket Properties** dialog box, which you use to view and modify the properties of the brackets that you are about to place in the model. The initial bracket

properties default from the selected structural specification. For more information, see *Bracket Properties Dialog Box* (on page 128).

🗋 Select Plane

Select the plane upon which you want to place the bracket.

🔲 Select Bracket Supports

Define the structural support parts for the bracket. You must create detail parts for the supports before you can specify them as bracket supports. For more information, see *Execute Detailing Command* (on page 54). Brackets require at least two, but not more than five, structural support parts.

⟷ Select Thickness Direction

Move the pointer to change the thickness direction for the bracket.

Finish

Places the bracket using the defined parameters.

Define Plane Controls

🔵 By Plane or Offset Plane

Places the plate system a specified distance from the reference plane. If you choose this option, you must define the offset distance. An offset distance of zero defines a coincident plane.

🗒 Plane by Point and Vector

Specifies the reference plane by defining a vector perpendicular to the reference plane. A third point defines the reference plane position along the vector.

🗋 Plane by Three Points

Specifies the reference plane using three points that you specify in the model.

🔒🔓 Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves. The **Lock** option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance for the plate part from the selected reference plane. You can specify the offset dynamically in graphics or by typing the distance. The **Offset** option is only available when you use the **Offset from a Plane** option.

Angle

Specify the angle at which to place the plate part relative to the reference plane. You have to define the axis of rotation using two points before you can define the angle. The **Angle** option is only available when you use the **Angle from plane** option.

Step

> Specifies the angle step. The step is incremented by this value when the cursor is moved in the graphic view. The **Step** option is only available when you use the **Angle from plane** option.

Select Vector

> Select or define a vector normal to the needed plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point

> Specify the point along the vector at which to place the plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point 1

> Specify the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 2

> Specify the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 3

> Specify the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Support 1

> Select the first support for the bracket.

Support 2

> Select the second support for the bracket.

Support 3

> Select the third support for the bracket.

Support 4

> Select the fourth support for the bracket.

Support 5

> Select the fifth support for the bracket.

Shape

> Select the shape of the bracket.

Cancel

> Clears all selected planes or boundaries.

Accept

> Accepts all selected planes or boundaries.

Parent System

Select the parent system for the plate system that you are placing. Systems are defined using the **Systems and Specifications** task.

Name

Displays the name of the plate system that you are placing.
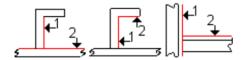
What do you want to do?

- *Create a two support bracket* (on page 124)
- *Create a three support bracket* (on page 125)
- *Create a four support bracket* (on page 126)
- *Create a five support bracket* (on page 127)
- *Modify a bracket* (on page 128)
- *Delete a bracket* (on page 128)

Create a two support bracket

1. Click **Place Bracket** .

2. On the **Place Bracket** ribbon, click **Select Plane** .

3. Define the plane upon which to place the bracket.

4. Click **Support 1** .

5. Select the surface port on the first profile or plate part.

6. Click **Support 2** .
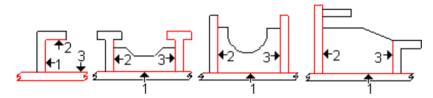
7. Select the surface port on the second profile or plate part.

8. If necessary, click **Properties** to define properties and defaults for the bracket.

9. Click **Select Thickness Direction** and specify the side of the definition plane on which to apply the bracket thickness.

   TIP When you select a surface port for supports, an arrow appears in the graphic view to show you which side of the support part you are selecting. Move the pointer to select the other side.

NOTE You must create a detail part for each profile part that you want to use as a bracket support. For more information, see *Execute Detailing Command* (on page 54).

Create a three support bracket
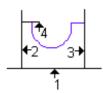
1. Click **Place Bracket** .
2. On the **Place Bracket** ribbon, click **Select Plane** .
3. Define the plane upon which to place the bracket.
4. Click **Support 1** .
5. Select the surface port on the first profile or plate part.
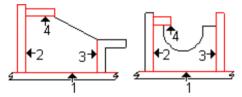
6. Click **Support 2** .
7. Select the surface port on the second profile or plate part.
8. Click **Support 3** .
9. Select the surface port on the third profile or plate part.
10. If necessary, click **Properties** to define properties and defaults for the bracket.
11. Click **Select Thickness Direction** and specify the side of the definition plane on which to apply the bracket thickness.

   TIP When you select a surface port for supports, an arrow appears in the graphic view to show you which side of the support part you are selecting. Move the pointer to select the other side.

NOTE You must create a detail part for each profile part that you want to use as a bracket support. For more information, see *Execute Detailing Command* (on page 54).

Create a four support bracket
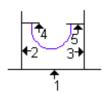
1. Click **Place Bracket** .

2. On the **Place Bracket** ribbon, click **Select Plane** .

3. Define the plane upon which to place the bracket.

4. Click **Support 1** .

5. Select the surface port on the first plate or profile part.

6. Click **Support 2** .

7. Select the surface port on the second profile or plate part.

8. Click **Support 3** .

9. Select the surface port on the third profile or plate part.
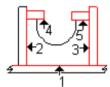
10. Click **Support 4** .

11. Select the surface port on the fourth profile or plate part.

12. If necessary, click **Properties** to define properties and defaults for the bracket.

13. Click **Select Thickness Direction** and specify the side of the definition plane on which to apply the bracket thickness.

   TIP When you select a surface port for supports, an arrow appears in the graphic view to show you which side of the support part you are selecting. Move the pointer to select the other side. No arrow appears for the last surface port, because there is only one valid solution.

NOTE You must create a detail part for each profile part that you want to use as a bracket support. For more information, see *Execute Detailing Command* (on page 54).

Create a five support bracket

1. Click **Place Bracket** .

2. On the **Place Bracket** ribbon, click **Select Plane** .

3. Define the plane upon which to place the bracket.

4. Click **Support 1** .

5. Select the surface port on the first plate or profile part.

6. Click **Support 2** .

7. Select the surface port on the second profile or plate part.

8. Click **Support 3** .

9. Select the surface port on the third profile or plate part.

10. Click **Support 4** .

11. Select the port on the fourth profile or plate part.

12. Click **Support 5** .

13. Select the surface port on the fifth profile or plate part.

14. If necessary, click **Properties** to define properties and defaults for the bracket.

15. Click **Select Thickness Direction** and specify the side of the definition plane on which to apply the bracket thickness.

    TIP When you select a surface port for supports, an arrow appears in the graphic view to show you which side of the support part you are selecting. Move the pointer to select the other side. No arrow appears for the last surface port, because there is only one valid solution.

NOTE You must create a detail part for each profile part that you want to use as a bracket support. For more information, see *Execute Detailing Command* (on page 54).

Modify a bracket

1. Click **Select** 🐾 on the vertical toolbar.

2. Select the bracket to modify.

   💡 **TIP** You can select the bracket in a graphic view or in the **Workspace Explorer**.

3. Using the ribbon controls, edit the bracket as needed.

💡 **TIPS**

▪ You must clear the **Rule Based** check box to change the item.

▪ If you select multiple detailed brackets of the same type, the **Item** box displays on the ribbon.

▪ If the shape is the same for each item, that value displays in the box and a list of other available shapes displays in the list. If you select a different shape, that shape is then applied to all of the items.

▪ If the shapes are not the same for all the selected items, then the box is blank and the list displays the available shapes. If you select a shape, that shape is then applied to all of the selected items. If you do not select a shape, the software creates the items according to the rule default values.

▪ If there are no common shapes, the **Item** box is unavailable and the software creates the items according to the rule default values.

📝 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a bracket

1. Click **Select** 🐾 on the vertical toolbar.

2. Select the bracket to delete.

3. Click **Delete** ✕.

Bracket Properties Dialog Box

Specifies the properties for the bracket that you are editing.

See Also

Main Tab (Bracket Properties Dialog Box)

Specifies the general properties of the bracket.

Name

Specifies the name of the plate part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the plate part. Select **User Defined** to type a name in the **Name** box.

Type

Specifies the type of plate part that you are placing. The default for a bracket part is **Bracket**.

Naming Category

Select a category for the plate part. The category specifies the role of the plate part in the model and is used by the naming rule to name the plate part.

Parent System

Specifies the parent plate system for the bracket part. The default for a bracket part is the leaf plate system of the first bracket support. You can change the parent system to the root plate system (the parent of the leaf plate system), the root or leaf plate system of another bracket support, or a system defined in the Systems and Specifications task. When the bracket part is created, the property values for the parent plate system are used for the initial bracket property values. When a parent property value changes, the corresponding bracket property value is not updated.

Surface Geometry Type

Displays **Planar Plate**, similar to the Molded Forms command used to create a planar plate system.

Specification

Select the structural specification for the plate part.

Description

Type a description for the plate part.

Tightness

Select the level of tightness as it applies to the entire plate part.

Board Management

Symmetry - Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Manually Override

Turn on this option to stop the Board Management Service from considering this part.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

See Also
Bracket Properties Dialog Box (on page 128)

Material Tab (Bracket Properties Dialog Box)

Specifies the material properties for the bracket that you are editing.

Material

Select the material for the plate part.

Grade

Select the material grade for the plate part.

Actual Thickness

Select the material thickness for the plate part.

See Also
Bracket Properties Dialog Box (on page 128)

Selection Tab (Bracket Properties Dialog Box)

Specifies the rule criteria used for bracket part selection.

? TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not

editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

    ◦ NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Results - Select the bracket class.

★ IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Bracket Properties Dialog Box (on page 128)

S E C T I O N   7

Placing Standalone Stiffener Profile Parts

The **Standalone Stiffener Profile Parts** commands define a single profile part or multiple profile parts.

Three methods are available for creating the landing curves of the stiffener profile parts:
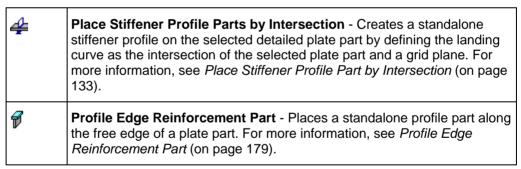
- By intersection
- By offset
- By two-dimensional sketch projection

When you create a landing curve by one of these methods, it could be linear, curved, polyline, circle, circular arc, or elliptical in shape.

You can use various input methods to create profile parts. You choose a method based on the type of trace curve that you need to create.

You can use any of the following methods to easily create profile parts for vertical, horizontal, slope, and curve:

- Typing the location line and the inclination angle from an orthogonal surface
- Specifying a distance from a reference surface or grid system
- Referring an existing surface and defining a nonorthogonal surface tangent to the existing surface (for example, the surface of a stiffener connected to a longitudinal frame).

	Place Stiffener Profile Parts by Intersection - Creates a standalone stiffener profile on the selected detailed plate part by defining the landing curve as the intersection of the selected plate part and a grid plane. For more information, see *Place Stiffener Profile Part by Intersection* (on page 133).
	Profile Edge Reinforcement Part - Places a standalone profile part along the free edge of a plate part. For more information, see *Profile Edge Reinforcement Part* (on page 179).

Place Stiffener Profile Part by Intersection

Creates a standalone stiffener profile part on the selected detailed plate part by defining the landing curve of the stiffener as the intersection of the selected plate part and a grid plane.

You can control the boundaries of the stiffener profile by selecting a plate system or part, profile system or part, seam, or grid plane that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the stiffener profile boundaries.

After defining the landing curve and the boundaries, you can set the profile orientation. If the profile part is placed on a plate system, the plate system side and the flange direction for unsymmetrical profile sections are set by the molded convention of the plate system. However, you can override the default settings with the **Position Definition** control on the ribbon. If the profile part is placed on a standalone plate part, the plate side and flange direction are toggled directly with the **Position Definition** control on the ribbon.

Stiffener Profile Part by Intersection Ribbon

Displays the options that you use to place a standalone stiffener profile part in the model.

Profile by Intersection Properties

Activates the **Profile Part Properties** dialog box, which you use to view and modify the properties of the profile part that you are about to place in the model. If the profile part is placed on a plate system, the initial settings default from the parent plate system. For more information, see *Profile Part Properties Dialog Box* (on page 136).

Plate System or Part

Select the plate system or part to stiffen. You can select multiple plate systems to place multiple stiffener profiles systems at the same time. The software uses the same information for each stiffener profile system. After placement, the stiffener profiles are unrelated to each other and can be edited as individual entities.

Section Type

Select the cross section type for the profile.

Section Size

Select the cross section size for the profile.

Name

Displays the name of the stiffener profile that you are placing.

Landing Curve Definition

Select the grid planes that intersect the plate system or part at which to place the stiffener profiles.

Position Definition

Select the plate side, mounting face, load point, and flange direction. See the **Position Definition** controls described below.

Position Definition

Flip Side

Defines the side of the plate system or profile part on which the profile part is placed. Clicking the control toggles the profile between the two sides.

Flip Direction

Defines the flange direction for an unsymmetrical profile cross section. Clicking the control toggles the flange between the two sides.

Mounting Face

Defines the mounting face of the profile part.

Load Point

Defines the load point of the profile part. Only the load points for the defined mounting face are displayed.

Boundary Definition

Define the boundaries of the stiffener profile. You can define the boundaries by selecting objects in the model. If you select objects in the model, and those objects are moved, the software automatically resizes the stiffener profile to maintain the boundary relationship.

Boundary Definition Controls

Pick Boundaries

Selects objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 97).

Boundary 1

Select the first boundary.

Boundary 2

Select the second boundary.

Finish

Places the plate system using the defined parameters.

Cancel

Clears all selected objects.

✅ **Accept**

Accepts all selected objects.

What do you want to do?

- *Place a stiffener profile part by intersection* (on page)
- *Modify a stiffener profile part* (on page)
- *Delete a stiffener profile part* (on page)

Place a stiffener profile part by intersection

1. Click **Stiffener Profile Part by Intersection** ⚓ on the vertical toolbar.
2. Select the plate system to stiffen.
3. Click **Accept** ✅.
4. Select a grid plane that intersects the plate system that you selected.
5. Click **Accept** ✅.
6. Define the side and flange direction of the stiffener profile with regard to the plate system.
7. Click **Accept** ✅.
8. Define the orientation of the stiffener profile with regard to the plate system.
9. Click **Accept** ✅.
10. Define the boundaries for the stiffener profile.
11. Click **Accept** ✅.
12. Click **Finish**.

📝 NOTE You can select more than one grid plane to place multiple stiffener profiles at one time.

Modify a stiffener profile part

1. Click **Select** 🖱 on the vertical toolbar.
2. Select **Stiffener Parts** in the **Locate Filter** box.
3. Select the stiffener profile to modify.
4. Click **Edit > Properties**.
5. Modify the stiffener profile as needed.
6. Click **OK**.

📝 NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a stiffener profile part

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Stiffener Parts** in the **Locate Filter** box.

3. Select the stiffener profile to delete.

4. Click **Delete** ✕.

Modify a beam profile part

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Beam Parts** in the **Locate Filter** box.

3. Select the linear beam profile to modify.

4. Click **Edit > Properties**.

5. Modify the profile as needed.

📝 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a beam profile part

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Beam Parts** in the **Locate Filter** box.

3. Select the beam profile part to delete.

4. Click **Delete** ✕.

Profile Part Properties Dialog Box

Specifies the settings for the profile part.

See Also
Main Tab (Profile Part Properties Dialog Box) (on page 136)
Cross Section Tab (Profile Part Properties Dialog Box) (on page 139)
Section Orientation Tab (Profile Part Properties Dialog Box) (on page 139)

Main Tab (Profile Part Properties Dialog Box)

Specifies the general settings for the profile.

Name

Displays the name that is assigned to the profile. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see Using Global Workshare in the *Global Workshare* Guide.

Rule

Specifies how the name is created. If set to a naming rule, the software automatically generates the name for you. If you select **User Defined**, you can type in the profile name yourself in the **Name** box.

Type

Select the type of profile. The profile type is used by the naming rule in naming the profile part.

Naming Category

Select a naming category. The category is used by the naming rule in naming the profile part that is a child to the profile system.

Parent System

Displays the name of the parent system of the profile.

Landing Curve Definition Method

Displays the method by which the landing curve is defined.

Specification

Displays the specification for the profile that was inherited from the parent system.

Description

Type a description for the profile.

The **Tools** > **Board Management Service** command determines structural part and seam symmetry about the center plane.

- Naming rules can consider symmetry information. For example, you can include a suffix indicating the symmetry property value for parts. Naming rules can also sort parts within a block by the symmetry value before assigning a part name index.

- The board property can be used in the nesting and cutting processes by possibly nesting and then cutting symmetrical parts at the same time, or nesting the symmetrical parts on the same raw material stock if the parts are in the same block. The Board Management Service determines if parts are symmetrically similar. Structural Manufacturing determines if the parts are symmetrically identical for manufacturing purposes.

- Unfolding algorithms use seam symmetry information when they calculate shapes for plates that cross the centerline.

- Some assembly planning tasks use the board property to automate creating and maintenance of assembly hierarchies for symmetrical parts.

- Reports include part symmetry information.

- Seam board information supports shell expansion drawing requirements. The shell expansion drawing for the entire ship displays and labels both port and starboard shell seams in one view.

- You can choose to manually override board management review for an object. If you choose this option, then board management disregards the object for subsequent review.

Whether you choose to run the Board Management service periodically or continuously depends upon your processes and workflow. If you need to see the symmetry immediately, then running this process continuously on a remote server may best suite your needs. If you do not need

immediate results, then you can run it periodically on your local computer at times that best fit your workflow.

▉ NOTE This property affects how Smart 3D names parts. It also affects how the parts are handled in Planning and Structural Manufacturing. Because of this, we recommend that you run the Board Management service before you generate drawings, bills of materials, or manufacturing output.

Board Management Service Console

Specifies the settings for running the board management service.

Server

Displays the name of the server on which the board management service runs.

Started by

Displays the user name of the person who started the board management service.

Seams Selected for Evaluation

Hull Seams

Indicates that seams on hull surfaces should be evaluated for symmetry.

Interior Seams

Indicates that seams on interior surfaces should be evaluated for symmetry.

Filters Applied to Target Selection

by Type

Indicates that parts should be filtered by type. The **Type** property is set on the **System Properties** dialog box.

by Naming Category

Indicates that parts should be filtered by naming category. The **Naming Category** property is set on the **System Properties** or **Part Properties** dialog box.

by Region

Indicates that parts should be filtered by region or bounding box. The bounding box is defined by the minimum and maximum x, y, and z locations of the object.

Growth

Specifies the amount to expand or shrink the region before filtering. The intent is to find parts that are approximately the same size and in approximately the same (mirrored) location.

Previous Part List Date/Time

Displays the date and time of the previous generation of the list of new and modified parts.

Reset Date/Time to

Replaces the start value of the last time the service was run. This time is used to search for new and modified parts.

Processing Status

Displays information about the number of parts processed and an estimation of the time remaining.

Start

Starts the board management service.

Stop

Quits the board management service without completing the process.

Close

Saves the values and closes the dialog box.

See Also
Profile Part Properties Dialog Box (on page 136)

Cross Section Tab (Profile Part Properties Dialog Box)

Specifies the cross section settings for the profile.

Material

Select the material from which the profile part is constructed.

Grade

Select the grade of the profile part material.

Section Type

Select the profile section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size

Select the profile section size.

See Also
Profile Part Properties Dialog Box (on page 136)

Section Orientation Tab (Profile Part Properties Dialog Box)

Specifies the position of the profile cross-section.

Mounting Face
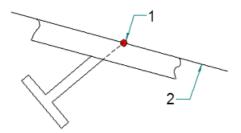
Select the cross-section face to mount to the plate system.

Load Point

Select the cross-section load point through which the landing curve passes.
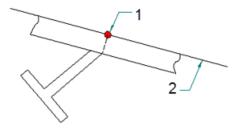
Landing Curve Projection Method

Select the projection method for the profile part.

1 - Landing Curve
2 - Molded Surface

Along Web Method (default)

1 - Landing Curve
2 - Molded Surface

Normal Method

Attachment Method

Select the method for attaching the profile part.

See Also
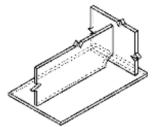
Profile Part Properties Dialog Box (on page 136)

SECTION 8

Place Corner Feature

Places a planar curve from the catalog at a selected corner. The software places the curve coplanar to the face of a part. The feature is applied to the part as a cutout. You can place corner features on plates, profiles, or members.

NOTE You can use the **Place Corner Feature** command to place corner features for drain or vent holes on clips generated as part of the **Execute Detailing** command.

Place Corner Feature Ribbon

Displays the controls used to place corner features.

Properties

Activates the **Corner Feature Properties** dialog box, which you use to view and modify the properties of the corner feature that you are about to place in the model. The most important properties for features are the rule selection criteria and the feature parameters. The initial corner feature properties default from the corner feature rules for the selected part type. For more information, see *Corner Feature Properties Dialog Box* (on page 143).

Corner Point

Select a plate, profile, or member part near the corner to place the corner feature.

Shape

Select the shape for the corner feature from the list. The list of available shapes is determined by the type of part selected and the rules you have defined.

NOTE During initial creation of the feature, click **Accept** to show selections in the **Shape** list.

Rule Based

Enables or disables the **Shape** list. Selecting this check box disables the **Shape** list.

Finish

Places the corner feature using the defined parameters.

Cancel

Clears the selected plate, profile, or member part.

☑ **Accept**

Accepts the selected plate, profile, or member part.

What do you want to do?

▪ *Place a corner feature* (on page)

▪ *Modify a corner feature* (on page 142)

▪ *Delete a corner feature* (on page 143)

Place a corner feature

1. Click **Place Corner Feature** ▢.

2. On the **Place Corner Feature** ribbon, click **Corner Point** 📐.

3. Select near a corner on the plate, profile, or member part to which to add the corner feature.

4. Click **Accept** ☑.

   ▤ **NOTE**   The **Rule Based** check box must be clear to select the corner feature shape.

5. Select the corner feature from the **Shape** list.

6. Click **Finish**.

7. If necessary, click **Properties** to define properties and defaults for the corner feature.

Modify a corner feature

1. Click **Select** 🔧 on the vertical toolbar.

2. Select **Features** in the **Locate Filter** box.

3. Select the corner feature to modify.

   ● **TIP** You can select the corner feature in a graphic view or in the **Workspace Explorer**.

4. Using the ribbon controls, edit the corner feature as needed.

   ● **TIPS**

   ▪ If you select multiple corner features of the same type, the **Item** box displays on the ribbon.

   ▪ If the shape is the same for each item, that value displays in the box and a list of other available shapes displays in the list. If you select a different shape, that shape is then applied to all of the items.

   ▪ If the shapes are not the same for all the selected items, then the box is blank and the list displays the available shapes. If you select a shape, that shape is then applied to all of the selected items. If you do not select a shape, the software creates the items according to the rule default values.

   ▪ If there are no common shapes, the **Item** box is disabled and the software creates the items according to the rule default values.

NOTE When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a corner feature

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Features** in the **Locate Filter** box.

3. Select the corner feature to delete.

4. Click **Delete** ✖.

Corner Feature Properties Dialog Box

Specifies the properties for the corner feature that you are editing.

See Also
Main Tab (Corner Feature Properties Dialog Box) (on page 143)
Selection Tab (Corner Feature Properties Dialog Box) (on page 144)
Configuration Tab (on page 401)

Main Tab (Corner Feature Properties Dialog Box)

Specifies the general properties of the corner feature part system.

Feature Name

Name

Specifies the name of the corner feature. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use. Select **User Defined** to specify the name yourself in the **Name** box.

See Also
Corner Feature Properties Dialog Box (on page 143)

Selection Tab (Corner Feature Properties Dialog Box)

Specifies the rule criteria used for corner feature selection.

♥ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  ⊟ NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Corner Placement

Select the corner placement method.

Arrest Stress Cracking

Select **Yes** if the corner feature is to provide relief from stress cracking at the corner.

Drainage on Part

Select **Yes** if the corner feature is to provide drainage.

Corner Feature Orientation

Select **NoFlip** if the corner is to use its default orientation along the local u and v axes of the corner. This only has an effect on non-symmetrical corner features.

Results

Select the corner feature class. Only valid selections based on user answers display.

Results

Select the corner feature. Only valid selections based on the user answers and selected corner feature class display.

Valid

Click after making any changes to the user answers so that valid results for the answers are displayed in the **Results** box. Also, click if the corner feature class changes in the first **Results** box so that only valid corner features are displayed in the second **Results** box. A preview of the corner feature displayed in the second **Results** field displays in the graphic view.

★ IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.
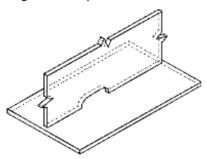
See Also
Corner Feature Properties Dialog Box (on page 143)

SECTION 9

Place Edge Feature

Places a planar curve from the catalog along a selected edge, at a selected point on the edge, and co planar to the face of a part. The feature is applied to the part as a cutout.

NOTE You can use the **Place Edge Feature** command to place edge features for drain or vent holes on clips generated as part of the **Execute Detailing** command.

Place Edge Feature Ribbon

Displays the controls used to place edge features.

Properties

Activates the **Edge Feature Properties** dialog box, which you use to view and modify the properties of the edge feature that you are about to place in the model. The most important properties for features are the rule selection criteria and the feature parameters. The initial edge feature properties default from the edge feature rules for the selected part type. For more information, see *Edge Feature Properties Dialog Box* (on page 150).

Select Edge

Select the edge of the plate or profile part to place the edge feature.

Select Position Point

Select the position on the edge of the plate or profile part to place the edge feature

Finish

Places the edge feature using the defined parameters.

Type

Select the type of edge feature to place. Available options include edge feature and waterstop.

Shape

Select the shape for the edge feature from the list. The list of available shapes is determined by the type of part that you have selected and the rules that you have defined.

Rule Based

Enables or disables the **Shape** list. Selecting this check box disables the **Shape** list.

Method

Select the placement method for the edge feature. Available options include offset, seam or knuckle, and point along edge.

The following controls are available if the **Method** field is set to **Offset**.

⬚ Reference

Select a structural object from which to measure the offset distance. This option is only available if you are using the offset method.

Direction

Select the direction in which to measure the offset distance from the reference. Available options include along edge, transverse, longitudinal, and vertical.

Flip

Reverses the offset direction from the reference.

Offset

Specifies the offset distance for the edge feature. After you type an offset distance, press ENTER or click ✅ **Accept** to display a preview.

✅ Accept

Accepts the inputs and computes a preview of the edge feature.

📝 **NOTE** During initial creation of the feature, click **Accept** to show selections in the **Shape** list.

What do you want to do?

- *Place an edge feature by offset* (on page)
- *Place a waterstop edge feature* (on page 148)
- *Place an edge feature at a seam or knuckle* (on page 149)
- *Place an edge feature at a point along an edge* (on page 149)
- *Modify an edge feature* (on page 150)
- *Delete an edge feature* (on page 150)

Place an edge feature by offset

1. Click **Place Edge Feature** .

2. On the **Place Edge Feature** ribbon, click **Select Edge** .

3. Select the edge of the plate or profile part to which to add the edge feature.

4. Click **Select Position Point** .

5. Set the **Method** box to **Offset**.

6. Select the direction method to use from the **Direction** list.

7. Type the offset distance from the reference position in the **Offset** box.

8. If necessary, click **Flip** to reverse the direction of the offset from the reference position.

9. Click **Accept** to display a preview of the edge feature.

10. Select **Edge Feature** from the **Type** list.

    NOTE The **Rule Based** check box must be clear to select the edge feature shape.

11. Select the edge feature shape from the **Shape** list.

12. Click **Finish**.

13. If necessary, click **Properties** to define properties and defaults for the edge feature.

Place a waterstop edge feature

1. Click **Place Edge Feature** .

2. On the **Place Edge Feature** ribbon, click **Select Edge** .

3. Select the edge of the plate or profile part to which to add the waterstop edge feature.

4. Click **Select Position Point** .

5. Select the point along the selected edge to be the center of the waterstop edge feature.

6. Select the positioning method to use from the **Method** list.

7. Select **Waterstop** from the **Type** list.

    NOTE The **Rule Based** check box must be clear to select the edge feature shape.

8. Select the shape to use for the waterstop from the **Shape** list.

9. Click **Finish**.

10. If necessary, click **Properties** to define properties and defaults for the edge feature.

Place an edge feature at a seam or knuckle

1. Click **Place Edge Feature** .

2. On the **Place Edge Feature** ribbon, click **Select Edge** .

3. Select the edge of the plate or profile part to which to add the edge feature.

4. Click **Select Position Point** .

5. Set the **Method** box to **Seam or Knuckle**.

6. Select the seam or knuckle at which to place the edge feature.

7. Select **Edge Feature** from the **Type** list.

    NOTE The **Rule Based** check box must be clear to select the edge feature shape.

8. Select the edge feature shape from the **Shape** list.

9. Click **Finish**.

10. If necessary, click **Properties** to define properties and defaults for the edge feature.

Place an edge feature at a point along an edge

1. Click **Place Edge Feature** .

2. On the **Place Edge Feature** ribbon, click **Select Edge** .

3. Select the edge of the plate or profile part to which to add the edge feature.

4. Click **Select Position Point** .

5. Set the **Method** box to **Point Along Edge**.

6. Select the point at which to place the edge feature.

7. Select **Edge Feature** from the **Type** list.

8. Select the edge feature shape from the **Shape** list.

    NOTE You must clear the **Rule Based** check box to select the edge feature shape.

9. Click **Finish**.

10. If necessary, click **Properties** to define properties and defaults for the edge feature.

Modify an edge feature

1. Click **Select** ![icon] on the vertical toolbar.

2. Select **Features** in the **Locate Filter** box.

    ![NOTE] The **Rule Based** check box must be clear to select the edge feature.

3. Select the edge feature to modify.

    ![TIP] You can select the edge feature in a graphic view or in the **Workspace Explorer**.

4. Using the ribbon controls, edit the edge feature as needed.

    ![TIPS]

    -   If you select multiple edge features of the same type, the **Item** box displays on the ribbon.

    -   If the shape is the same for each item, that value displays in the box and a list of other available shapes displays in the list. If you select a different shape, that shape is then applied to all of the items.

    -   If the shapes are not the same for all the selected items, then the box is blank and the list displays the allowed available shapes. If you select a shape, that shape is then applied to all of the selected items. If you do not select a shape, the software creates the items according to the rule default values.

    -   If there are no common shapes, the **Item** box is disabled and the software creates the items according to the rule default values.

Delete an edge feature

1. Click **Select** ![icon] on the vertical toolbar.

2. Select **Features** in the **Locate Filter** box.

3. Select the edge feature to delete.

4. Click **Delete** ![icon].

Edge Feature Properties Dialog Box

Specifies the properties for the edge feature that you are editing.

See Also

Main Tab (Edge Feature Properties Dialog Box) (on page 150)
Selection Tab (Edge Feature Properties Dialog Box) (on page 151)
Parameter Tab (on page 153)
Members Tab (on page 171)
Controlled Interfaces Tab (Edge Treatment Properties Dialog Box) (on page 171)
Configuration Tab (on page 401)
General Tab (on page 402)

Main Tab (Edge Feature Properties Dialog Box)

Specifies the general properties of the edge feature.

Feature Name

Name

Specifies the name of the edge feature. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use. Select **User Defined** to specify the name yourself in the **Name** box.

Board Management

Symmetry

Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Manually Override

Select this option to stop the Board Management Service from considering this part.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

See Also
Edge Feature Properties Dialog Box (on page 150)
Board Management Service (on page 380)

Selection Tab (Edge Feature Properties Dialog Box)

Specifies the rule criteria used for edge feature selection.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Edge Placement

Select the edge placement method.

Off of Edge

Select **Yes** if the edge feature is to be offset from the edge. Select **No** if the edge feature goes to the edge.

Drainage on Part

Select **Yes** if the edge feature is to provide drainage.

Apply Treatment

Select **Yes** to apply an edge treatment to the feature.

Results

Select the edge feature class. Only valid selections based on the user answers display in the list.

Results

Select the edge feature. Only valid selections based on the user answers and the selected edge feature class display in the list.

IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Edge Feature Properties Dialog Box (on page 150)

Parameter Tab

Specifies the parameters for the feature.

Parameter

Displays the name of the parameter.

Value

Displays the value of the parameter. Values that are grayed cannot be modified.

Rule-Based

Displays a checkbox if the parameter value is rule-based. Clear the check box to modify the value in the **Value** box.

Catalog Value

Displays the original catalog value if the **Value** box has been modified.

See Also
Corner Feature Properties Dialog Box (on page 143)
Edge Feature Properties Dialog Box (on page 150)

S ECTION 10

Place Edge Features at Seams

Places a planar curve from the catalog where the edge of the part meets a seam. The feature is applied to the part as a cutout.

Place Edge Feature at Seams Ribbon

Displays the controls used to place edge features at seams.

Select Plate or Profile

Select the plate or profile to add the edge features.

Finish

Places edge features on the select objects at all intersecting seams.

Cancel

Clears the selected edge.

Accept

Accepts the selected edge.

Shape

Select the shape for the edge feature from the list. The list of available shapes is determined by the type of part that you have selected and the rules that you have defined.

Rule Based

Enables or disables the **Shape** list. Selecting this check box disables the **Shape** list.

NOTES

- During initial creation of the feature, click **Accept** to show selections in the **Shape** list.

- If the shape is the same for each edge feature, that value displays in the box and a list of other available shapes displays in the list. If you select a different shape, that shape is then applied to all of the selected edge features.

- If the shapes are not the same for all the selected edge features, then the box is blank and the list displays the available shapes. If you select a shape, that shape is then applied to all of the selected edge features. If you do not select a shape, the software creates the edge features according to the rule default values.

- If there are no common shapes, the **Shape** box is disabled and the software creates the edge features according to the rule default values.

Place an edge feature at seams

1. Click **Place Edge Feature at Seams** .

2. Select the plate or profile part to which to add the edge features.

3. Click **Accept** .

4. Click **Finish**.

Place Edge Features by Offset

 Places a series of planar curves from the catalog along a selected edge coplanar to the face of a part. You specify the number of edge features and the distance between them. The features are applied to the part as a cutout.

Place Edge Features by Offset Ribbon

Displays the controls used to place edge features by offset.

Select Edges

Select the edge of the plate or profile part at the location for the edge feature.

Select Reference

Select the position from which to start calculating the edge features. This point can be an object in the design or a point.

Define Offsets

Specify the distance between the edge features.

Unlimited

Specifies the same distance between all edge features. For more information, see *Offset Unlimited Case Dialog Box* (on page 158).

Table

Specifies unique distances between the features.

Shape

Select the shape for the edge feature from the list. The list of available shapes is determined by the type of part that you have selected and the rules that you have defined.

Rule Based

Enables or disables the **Shape** list. Selecting this check box disables the **Shape** list.

NOTES

▪ During initial creation of the feature, click **OK** on the **Unlimited** or **Table** dialog boxes to show selections in the **Shape** list. To change the feature shape you must clear the **Rule Based** check box.

- If the shape is the same for each edge feature, that value displays in the box and a list of other available shapes displays in the list. If you select a different shape, that shape is then applied to all of the selected edge features.

- If the shapes are not the same for all the selected edge features, then the box is blank and the list displays the available shapes. If you select a shape, that shape is then applied to all of the selected edge features. If you do not select a shape, the software creates the edge features according to the rule default values.

- If there are no common shapes, the **Shape** box is disabled and the software creates the edge features according to the rule default values.

Finish

Places the bracket using the defined parameters.

Cancel

Clears the current selection.

Accept

Accepts the current selection.

What do you want to do?

- *Place equally spaced edge features* (on page 157)

- *Place multiple edge features by table* (on page 158)

Place equally spaced edge features

1. Click **Place Edge Features by Offset** .

2. On the **Place Edge Feature by offset** ribbon, click **Select Edge** .

3. Select the edge of the plate or profile part to which to add the edge features.

4. Click **Select Start Reference Point** .

5. Select the point along the selected edge to be the reference point for the edge features.

6. Click **Accept** .

7. Click **Unlimited Case** .

8. Specify the number of edge features to place using the **+ve** or -**ve Count** boxes.

9. Specify the distance between the edge features using the **+ve** or -**ve Count** boxes.

10. Specify whether or not to use the reference start point as the location for the first edge feature.

11. Click **OK**.

    NOTE The **Rule Based** check box must be clear to select the edge feature shape.

12. Select the edge feature shape from the **Shape** list.

13. Click **Finish**.

Place multiple edge features by table

1. Click **Place Edge Features by Offset** .

2. On the **Place Edge Feature by offset** ribbon, click **Select Edge** .

3. Select the edge of the plate or profile part to which to add the edge features.

4. Click **Select Start Reference Point** .

5. Click **Accept** .

6. Click **By Table** .

7. Specify the number of features to place and the distance between each edge feature.

8. Click **OK**.

    NOTE The **Rule Based** check box must be clear to select the edge feature shape.

9. Select the edge feature from the **Shape** list.

10. Click **Finish**.

Offset Unlimited Case Dialog Box

Displays the controls used to place unlimited edge features.

+ve Offset

Displays offset distance in the positive direction from the reference point. The software displays an arrow along the selected edge to indicate the positive direction.

+ve Count

Displays the number of edge features to place in the positive direction.

-ve Offset

Displays offset distance in the negative direction from the reference point.

-ve Count

Displays the number of edge features to place in the negative direction.

Definition

Specifies how the distances are calculated.

Place Feature At Reference Point

Places an edge feature at the reference start point.

See Also
Place Edge Features by Offset (on page 156)

Offset Table for Edge Feature Dialog Box

Displays the controls used to place edge features by table.

Sequence

Displays the number of the edge feature, starting from the reference start point.

CoordSys

Displays the name of the coordinate system.

Reference

Displays the name of the reference point.

Offset

Displays the distance from the offset point.

Count

Displays the number of edge features to place.

Relative

Indicates that the offset distance is from the previous edge feature rather than the reference start point.

Insert Row Above

Adds a row to the table above the highlighted row.

Insert Row Below

Adds a row to the table below the highlighted row.

Delete Row

Removes the highlighted row.

Place Feature At Reference Point

Places an edge feature at the reference start point.

Definition

Specifies how the distances are calculated.
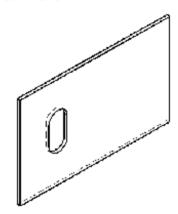
See Also

Place Edge Features by Offset (on page 156)

Place Sketched Features

 Places a closed planar curve, either from the catalog or sketched, coplanar to the face of a part. The feature is applied to the part as a cutout, similar to a hole. You can place sketched features on plates, profiles, or members. Both built-up and rolled members are supported by this command.

NOTES

- You can use the **Place Sketched Feature** command to place sketched features for drain or vent holes on clips generated as part of the **Execute Detailing** command.

- When you create or edit a sketched feature, the dialog box title displays the plate part name using the format **Plate Part:** *<name of the plate part>*.

- The **Place Sketched Feature** command can be used to place features that cut a corner or an edge of a part by sketching the feature contour so that it extends past the corner of the edge. The feature is not relational to the corner or the edge, so if the shape of the part is changed, the feature may disappear.

- On a curved plate, profile, member, or any other part where the feature could cut the part in more than one distinct location, the **Placed Sketched Feature** command only cuts the feature on the closest element to the sketch location. For example, if you want to cut both the top and bottom flange of an I-beam, two sketched features must be placed.

When you place a sketched feature, you can define opening properties to better reflect the more granular details of the purpose of the opening or hole, when it is cut, and how it is produced. These properties are grouped into the following categories:

Design Properties

Represent the functional purpose of the opening or hole. The design property reflects why the opening exists in the model. Typical design properties include:

- Access (Permanent)

- Access (Temporary)
- Lightening (Weight Reduction)
- Air Escape
- Liquid Escape
- Air/Liquid Escape
- Foothold
- Martyr

Planning Properties

Represent the staging of the opening or hole. The planning property denotes the point in the design and production lifecycle when the cut is physically made. Typical planning properties include:

- Design
- Shop (Assembly)
- Ship (Field)

Production Properties

Represent the production method of the opening or hole. The planning property denotes whether the cut is truly cut from the system or part, whether it is marked, bridged, and so forth. Typical production properties include:

- Cut (Default)
- Mark
- Bridge
- Chill Hole (Patch)

Place Sketched Features Ribbon

Specifies the properties for the opening that you are placing.

Sketched Feature Properties

Activates the **Sketched Feature Properties** dialog box, which is used to define properties of the opening that are not available on the ribbon, mainly the catalog of parametric sketched features. For more information, see *Sketched Feature Properties Dialog Box* (on page 445).

Select Plate or Profile Part

Select the plate, profile, or member part in which to place the opening.

Select Sketching Plane

Define the shape of the opening.

Finish

Places the plate system using the defined parametric catalog shape or sketched curve.

Sketching Plane

Specify the sketching plane for the opening. This is the first step in defining the opening.

Add Intersecting Item

Allows you to select objects that intersect the sketching plane in the 3D environment. You see the selected objects in the Sketch 2D environment when you are drawing the opening.
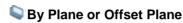
Add Projection Item

Allows you to select objects that do not intersect the sketching plane in the 3D environment. The objects are projected onto the sketching plane, and you see the selected objects in the Sketch 2D environment when you are drawing the opening.

Sketch 2D

Activates the Sketch 2D environment in which you can draw the opening.

By Plane or Offset Plane

Places the plate system a specified distance from the reference plane. If you choose this option, you must define the offset distance. An offset distance of zero defines a coincident plane.

Plane by Point and Vector

Specifies the reference plane by defining a vector perpendicular to the reference plane. A third point defines the reference plane position along the vector.

Plane by Three Points

Specifies the reference plane using three points that you specify in the model.

Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves. The **Lock** option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance for the plate part from the selected reference plane. You can specify the offset dynamically in graphics or by typing the distance. The **Offset** option is only available when you use the **Offset from a Plane** option.

Angle

Specify the angle at which to place the plate part relative to the reference plane. You have to define the axis of rotation using two points before you can define the angle. The **Angle** option is only available when you use the **Angle from plane** option.

Step

Specifies the angle step. The step is incremented by this value when the cursor is moved in the graphic view. The **Step** option is only available when you use the **Angle from plane** option.

Select Vector

Select or define a vector normal to the needed plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point

Specify the point along the vector at which to place the plane. This option is only available when you use the **Plane by Point and Vector** option.

Define Point 1

Specify the location of the first of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 2

Specify the location of the second of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Define Point 3

Specify the location of the third of three points that defines the plane. This option is only available when you are using the **Plane by Three Points** option.

Auto

Indicates whether Smart 3D automatically adds all related objects to the select set.

Cancel

Clears the current selection.

Accept

Accepts the current selection.

Opening design type

Specifies the design opening property for the opening or hole. The default value is **Permanent Access**. The list is defined by the CutoutDesignType codelist.

Opening planning method

Specifies the planning opening property for the opening or hole. The default value is **Design**. The list is defined by the CutoutPlanningMethod codelist.

Opening production method

Specifies the production opening property for the opening or hole. The default value is **Cut**. The list is defined by the CutoutProductionMethod codelist.

NOTE For more information about sketched features opening properties, see *Place Sketched Features* (on page 160).

What do you want to do?

- *Place a sketched feature by sketching* (on page 164)
- *Place a sketched feature from the catalog* (on page 165)
- *Modify a sketched feature* (on page 166)
- *Delete a sketched feature* (on page 166)

Place a sketched feature by sketching

1. Click **Place Sketched Feature** on the vertical toolbar.

2. Select the plate, profile, or member part in which to place the sketched feature.

3. Specify the sketching plane for the opening using one of the following methods:

   Define plane using angle from plane (on page 29)

   Define a coincident plane (on page 29)

   Define an offset plane (on page 30)

   Define a plane using three points (on page 31)

   Define a plane using a point and a normal vector (on page 32)

   Define plane by vectors normal (on page 33)

   ## ♥ TIPS

   - If you selected a planar plate part, its surface is automatically specified as the coincident sketch plane. You can specify another sketch plane by one of the other methods if needed.

   - If you selected a profile or member part, one of the surface ports on the web or flange can be specified as the sketch plane.

4. Click **Add Intersecting Item** .

5. Select objects in the Model to use as reference geometry that intersect the sketching plane. These extra objects appear in the Sketch 2D environment.

6. Click **Add Projection Item** .

7. Select objects in the Model to use as reference geometry that do not intersect the sketching plane. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

8. Click **Sketch 2D** .

9. Using the available drawing commands in the Sketch 2D environment, draw the sketched feature.

   ★ IMPORTANT You must sketch a closed contour.

10. Click **Finish** on the ribbon in the Sketch 2D environment.

11. Use the **Opening design type**, **Opening planning method**, and **Opening production method** lists to set the design, planning, and production properties on the sketched feature.

12. Click **Finish**.

Place a sketched feature from the catalog

1. Click **Place Sketched Feature** on the vertical toolbar.

2. Select the plate, profile, or member part in which to place the sketched feature.

3. Click **Properties** .

4. Specify the sketching plane for the opening using one of the following methods:

   Define plane using angle from plane (on page 29)

   Define a coincident plane (on page 29)

   Define an offset plane (on page 30)

   Define a plane using three points (on page 31)

   Define a plane using a point and a normal vector (on page 32)

   Define plane by vectors normal (on page 33)

   ## ☀ TIPS

   - If you selected a planar plate part, its surface is automatically specified as the coincident sketch plane. You can specify another sketch plane by one of the other methods if needed.

   - If you selected a profile or member part, one of the surface ports on the web or flange can be specified as the sketch plane.

5. Click **Add Intersecting Item** .

6. Select objects in the Model that intersect the sketching plane that you want to use as reference geometry. These extra objects will appear in the Draft 2D environment.

7. Click **Add Projection Item** .

8. Select objects in the Model to use as reference geometry that do not intersect the sketching plane. These extra objects are projected onto the sketching plane and appear in the Draft 2D environment.

9. Click **Sketch 2D** .

   ★ IMPORTANT The **Sketched Feature Properties** dialog box must be open when you click **Sketch 2D**. It cannot be accessed after the Draft 2D environment is open.

10. Select the **Catalog** tab on the **Sketched Feature Properties** dialog box.

11. Select the opening to place from the opening shapes in the catalog.

12. Drag the preview of the selected opening shape into the Draft 2D environment.

13. Click **OK** on the **Sketched Feature Properties** dialog box.

14. Click **Finish** on the ribbon in the Draft 2D environment.

15. Click **Finish**.

Modify a sketched feature

1. Click **Select** 🖱 on the vertical toolbar.

2. Select **Features** in the **Locate Filter** box.

3. Select the sketched feature to modify.

    💡 **TIP** You can select the sketched feature in a graphic view or in the **Workspace Explorer**.

4. Using the ribbon controls, edit the sketched feature as needed.

📝 **NOTES**

- When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

- If you select multiple sketched features from the same sketch file, you can edit the geometry for those features as well as the properties. If the sketched features are from multiple sketch files, then you can only edit the properties of those features.

Delete a sketched feature

1. Click **Select** 🖱 on the vertical toolbar.

2. Select the sketched feature to delete.

3. Click **Delete** ✖.

Place Edge Treatment

 Specifies the characteristics of free edges. Free edges often need to be shaped with a grinder, or beveled to satisfy paint adhesion, stress concentration or weld joint requirements.

A free edge is an edge that has no connection. That means that a free edge can be an opening edge (created in the Molded Forms task), a hole edge (created in the Hole Management task), an edge made by use of a boundary curve, the edge of a lapped plate, or an edge defined by use of a reference plane. A free edge treatment is any operation performed on a free edge, such as grind, bevel, or mill.

Place Edge Treatment Ribbon

Displays the controls used to place an edge treatment.

Properties

Activates the **Edge Treatment Properties** dialog box, which you use to view and modify the properties of the free edge treatment that you are about to place in the model. The most important properties for edge treatments are the rule selection criteria and the treatment parameters. The initial edge treatment properties default from the edge treatment rules for the selected part type (plate, profile, or member). For more information, see *Edge Treatment Properties Dialog Box* (on page 169).

Select Edge

Select the free edge on which to place the edge treatment.

Boundary 1

Select the position on the free edge to start the free edge treatment.

Boundary 2

Select the position on the free edge to stop the free edge treatment.

Finish

Places the edge treatment using the defined parameters.

Accept

Accepts the selected edge treatment.

Bound By

Specifies whether the free edge treatment is bound by an object or a point.

Type

Specifies the type of free edge treatment to apply.

Rule Based

Enables or disables the **Type** list. Selecting this check box disables the **Type** list.

What do you want to do?

- *Place a free edge treatment* (on page 168)
- *Modify a free edge treatment* (on page 168)
- *Copy a free edge treatment* (on page 169)
- *Delete a free edge treatment* (on page 169)

Place a free edge treatment

1. Click **Place Edge Treatment** ✏.
2. Select the free edge to which to add the free edge treatment.
3. Click **Accept** ☑ to execute the free edge treatment rules.

   📄 NOTE The **Rule Based** check box must be clear to select the edge treatment type.

4. Select the type of free edge treatment to apply from the **Type** list.
5. To place the free edge treatment along the entire length of the selected free edge, click **Finish**.
6. To specify a start point for the free edge treatment, click **Boundary 1** ↘.
7. Select the boundary method from the **Bound By** list.
8. Select the point or object to use for the start point of the free edge treatment.
9. To specify an end point for the free edge treatment, click **Boundary 2** ↘.
10. Select the boundary method from the **Bound By** list.
11. Select the point or object to use for the end point of the free edge treatment.
12. Click **Finish**.
13. If necessary, click **Properties** to define properties and defaults for the free edge treatment.

Modify a free edge treatment

📄 NOTE The **Rule Based** check box must be clear to select the edge treatment type.

1. Select the free edge treatment to modify.
2. Click **Properties** 🖼 to define properties and defaults for the free edge treatment.

💡 TIPS

- If you select multiple free edge treatments of the same type, the **Item** box displays on the ribbon.
- If the type is the same for each item, that value displays in the box and a list of other available types displays in the list. If you select a different type, that type is then applied to all of the items.

- If the types are not the same for all the selected items, the box is blank and the list displays the available types. If you select a type, that type is then applied to all of the selected items. If you do not select a type, the software creates the items according to the rule default values.

- If there are no common types, the **Item** box is disabled and the software creates the items according to the rule default values.

Copy a free edge treatment

1. Select the free edge treatment to copy.

2. Click **Edit > Copy**.

3. Select the free edge to apply the copied free edge treatment.

4. Click **Edit > Paste Special**.

5. If necessary, click **Properties** 🖳 to define properties and defaults for the free edge treatment.

Delete a free edge treatment

1. Select the free edge treatment to delete.

2. Click **Delete** ✕.

Edge Treatment Properties Dialog Box

Specifies the properties for the free edge treatment that you are editing.

See Also
Main Tab (Edge Treatment Properties Dialog Box) (on page 169)
Selection Tab (Edge Treatment Properties Dialog Box) (on page 170)
Bevel Parameters Tab (on page 171)
Members Tab (on page 171)
Controlled Interfaces Tab (Edge Treatment Properties Dialog Box) (on page 171)

Main Tab (Edge Treatment Properties Dialog Box)

Specifies the general properties of the edge feature.

Feature Name

Name

Specifies the name of the edge treatment. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use. Select **User Defined** to specify the name yourself in the **Name** box.

Type

Specifies the type of free edge treatment.

Length

Specifies the length of the free edge treatment.

See Also

Edge Treatment Properties Dialog Box (on page 169)

Selection Tab (Edge Treatment Properties Dialog Box)

Specifies the rule criteria used for free edge treatment selection.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default questions, answers, and results are determined by the rule class.

Edge Treatment

Select the edge treatment type.

Results

Select the edge treatment type. Only valid selections based on the user answers and the selected edge treatment class display in the list.

Grind Method

Select the method to apply.

Flip

Specify whether or not to flip the edge treatment.

Results

Select the edge treatment. Only valid selections based on the user answers and selected grind method display in the list.

★ IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Edge Treatment Properties Dialog Box (on page 169)

Bevel Parameters Tab

Specifies the bevel parameters used for the smart occurrence object, such as a physical connection or free edge treatment.

Reference Part

Specifies the part containing the bevel.

Depth

Specifies the depth of the bevels. The **Depth** boxes correspond to the dimensions on the preview picture.

Angle

Specifies the angle of the bevels. The **Angle** boxes correspond to the dimensions on the preview picture.

Members Tab

This tab is not applicable to free edge treatments.
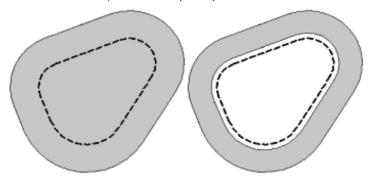
Controlled Interfaces Tab (Edge Treatment Properties Dialog Box)

This tab is not applicable to free edge treatments.

SECTION 14
Plate Edge Reinforcement Part

🟧 Places a standalone lapped plate part around a closed free edge (such as an opening or sketched feature) of another plate part.

You complete three steps when you are creating a plate edge reinforcement. First, use the **Properties** 🔲 command to define properties and defaults for the plate edge reinforcement. Next, define the edge to be reinforced. Then, define the shape of the plate by defining inner and outer offsets and the side of the reinforced plate on which to place the edge reinforcement.

Plate edge reinforcements can be solid (single offset) or ringed (double offset).

Plate Edge Reinforcement Part Ribbon

Displays the options that you use to place the plate edge reinforcement in the model.

🔲 **Plate Edge Reinforcement Properties**

Activates the **Plate Edge Reinforcement Part Properties** dialog box, which you use to view and modify the properties of the plate part that you are about to place in the model. The initial properties default from the selected structural specification. For more information, see *Plate Edge Reinforcement Part Properties Dialog Box* (on page 441).

👆 **Pick Edge**

Select the closed free edge to be reinforced. You can select only one edge.

👆 **Shape Definition**

Define the inner and outer contour offsets of the plate edge reinforcement.

🖊 **Side of Plate**

Define the side of the reinforced plate on which to place the plate edge reinforcement.

Finish

Places the reinforcement using the defined parameters.

❌ **Cancel**

Clears all selected planes or boundaries.

☑ **Accept**

Accepts all selected planes or boundaries.

Thickness Direction

Select the direction in which to apply the thickness. The software displays an arrow indicating the active direction.

Material

Select the material for the plate part.

Grade

Select the material grade for the plate part.

Thickness

Select the material thickness for the plate part.

Shape Definition Controls

Inner Offset

Specify the offset distance for the inner contour of the plate reinforcement from the selected edge.

💡 **TIPS**

- Pick a value of **Blank** for a solid reinforcement covering the opening or sketched feature.

- Type a value of **0** for a reinforcement flush with the edge.

- Type a negative value for a reinforcement overlapping the edge.

Outer Offset
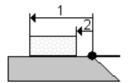
Specify the offset distance for the outer contour of the plate reinforcement from the selected edge. The outer offset must be a positive value.

What do you want to do?

- *Place a plate edge reinforcement part* (on page 174)

- *Modify a plate edge reinforcement part* (on page 174)

- *Delete a plate edge reinforcement part* (on page 174)

Place a plate edge reinforcement part

1. Click **Plate Edge Reinforcement Part** [icon] on the vertical toolbar.

2. Select the free edge to reinforce.

3. Specify the offsets to use.

   - 1 is the outer offset.

   - 2 is the inner offset.

4. Click **Finish**.

Modify a plate edge reinforcement part

1. Click **Select** [icon] on the vertical toolbar.

2. Select **Plate Parts** in the **Locate Filter** box.

3. Select the plate edge reinforcement part to modify.

4. Use the commands on the ribbon to make the necessary changes.

[NOTE icon] **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a plate edge reinforcement part

1. Click **Select** [icon] on the vertical toolbar.

2. Select **Plate Parts** in the **Locate Filter** box.

3. Select the plate edge reinforcement part to delete.

4. Click **Edit > Delete**.

See Also
Place Sketched Features (on page 160)

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Edge Reinforcement Properties Dialog Box

Specifies the settings for the profile edge reinforcement.

See Also
Main Tab (Edge Reinforcement Properties Dialog Box) (on page 175)
Cross Section Tab (Edge Reinforcement Properties Dialog Box) (on page 176)
Section Orientation Tab (Edge Reinforcement Properties Dialog Box) (on page 177)
Configuration Tab (on page 401)

Main Tab (Edge Reinforcement Properties Dialog Box)

Specifies the general settings for the profile edge reinforcement.

Name

Displays the name that is assigned to the part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Specifies how the name is created. If set to a naming rule, the software automatically generates the name for you. If you select **User Defined**, you can type the part name in the **Name** box.

Type

Displays the object type, which in this case is **Edge Reinforcement**.

Naming Category

Select a category for the part. Categories specify the role of the part in the model. The category is also used by the naming rule to name the part.

Parent System

Displays the name of the parent system of the part.

Description

Type a description for the edge reinforcement profile.

Specification

Select a specification for the edge reinforcement profile.

Symmetry

Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

See Also
Edge Reinforcement Properties Dialog Box (on page 175)

Cross Section Tab (Edge Reinforcement Properties Dialog Box)

Specifies the cross section settings for the edge reinforcement profile.

Section Type

Specifies the profile cross-section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size

Specifies the profile section size.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

The tab also displays the graphic and parameters of the cross section symbol that you have selected to use for the edge reinforcement profile.

Identifying Parent Values for Leaf Properties

For a property on a leaf system, the value assigned to the root system contains an asterisk (*), such as:

This makes it easy to see whether the currently assigned value for a leaf property is different from the parent property. You can also change a modifiable leaf property back to the root value without first checking the root value in the root system **Properties** dialog box.

🗒 NOTE The asterisk (*) only displays in the **Properties** dialog box for a leaf system.

Section Orientation Tab (Edge Reinforcement Properties Dialog Box)

Specifies the position of the cross-section relative to the free edge.

Position

Defines how the cross section of the profile is attached to the free edge. Your options are:

- **On Edge - Centered** - The cross-section is centered on the plate thickness using the cross-section load point that you select.

- **On Edge - Offset** - The cross-section load point is offset from the free edge the distance you specify in the **Offset** box.

- **On Face** - The cross-section load point is placed on the face of the plate. The load point can be offset from the free edge the distance you specify in the **Offset** box.
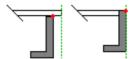
Primary Orientation

Specifies on which side of the plate system the profile is placed. The default setting is the molded conventions of the parent system.

Secondary Orientation
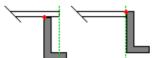
Specifies the orientation of the web thickness for symmetrical cross-sections and both the web thickness and the flange direction for unsymmetrical cross-sections. Your options are:

- **Toward Plate** - The plate edge is moved back so that the flange is at the plate edge.

- **From Plate** - The flange is added to the plate edge.

Parent System Type
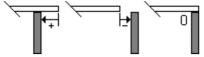
Displays the object type of the parent system. You cannot change this value.

Offset

Specify the distance that the cross-section is offset from free edge or from the plate face.

If **Position** is set to **On Face**, then the offset is measured from the free edge. A positive

offset value moves the cross- section towards the plate. A negative offset value moves the cross-section away from the plate.

If **Position** is set to **On Edge**, then the offset is measured from the plate face. A positive offset value moves the cross-section in the opposite direction from the primary orientation. A negative value moves the cross-section in the same direction as the primary orientation.

Mounting Face

Select the cross-section face to mount to the plate system.

Load Point

Select the cross-section load point through which the free edge passes.

See Also
Edge Reinforcement Properties Dialog Box (on page 175)

Profile Edge Reinforcement Part

Places a standalone profile part along the free edge of a plate part. A free edge is defined as a plate edge that has no logical connections. Therefore, a free edge can be a hole edge, an edge made by use of a boundary curve, or an edge defined by use of a grid plane.

Profile Edge Reinforcement Part Ribbon

Displays the options that you use to place a profile edge reinforcement part in the model.

Edge Reinforcement Properties

Activates the **Plate Edge Reinforcement Part Properties** dialog box, which you use to view and modify the properties of the edge reinforcement system that you are about to place in the model. For more information, see *Plate Edge Reinforcement Part Properties Dialog Box* (on page 441).

Pick Edge

Select the plate edge to reinforce. You can select multiple edges to place multiple edge reinforcements at the same time. The software uses the same boundaries and orientation for each edge reinforcement. After placement, the edge reinforcements are unrelated to each other and can be edited as individual entities.

Define Orientation

Specify the orientation of the edge reinforcement.

Boundary 1

Define the start point for the boundary of the edge reinforcement.

Boundary 2

Define the end point for the boundary of the edge reinforcement.

Finish

Places the edge reinforcement using the defined parameters.

Rule

Specifies the rule Smart 3D uses to place the edge reinforcement part.

Cancel

Clears all selected edges or boundaries.

Accept

Accepts all selected edges or boundaries.

Section Type

Select the profile cross-section type.

Section Size

Select the profile cross-section size.

Name

Displays the name of the edge reinforcement that you are placing.

What do you want to do?

- *Place an edge reinforcement profile part* (on page)
- *Modify an edge reinforcement profile part* (on page)
- *Delete an edge reinforcement profile part* (on page)

Place an edge reinforcement profile part

1. Click **Profile Edge Reinforcement Part** 🔳 on the vertical toolbar.
2. Select the edge to reinforce.
3. Click **Accept** ✅.
4. Define the orientation of the profile.
5. Click **Accept** ✅.
6. Define the boundaries for the profile.
7. Click **Accept** ✅.
8. Click **Finish**.

Modify an edge reinforcement profile part

1. Click **Select** 🔍 on the vertical toolbar.
2. Select **Profile Edge Reinforcement Parts** in the **Locate Filter** box.
3. Select the edge reinforcement profile part to modify.
4. Use the commands on the ribbon to modify the profile part as needed.

🗏 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete an edge reinforcement profile part

1. Click **Select** on the vertical toolbar.

2. Select **Profile Edge Reinforcement Parts** in the **Locate Filter** box.

3. Select the edge reinforcement profile part to delete.

4. Click **Edit > Delete**.

Place Multiple Corner Features

⬤ Places planar curves from the catalog at the corners of a selected part. The software places the curves coplanar to the face of a part. The features are applied to the part as a cutout.

You can place corner features on plate systems, plate parts, assemblies, and blocks.

Place Multiple Corner Features Ribbon

Displays the controls used to place multiple corner features.

⬤ Place Multiple Corner Features

Select a plate or profile part on which to place the corner features. You can place the corner features on multiple parts by selecting plate systems, plate parts, assemblies, or blocks.

Finish

Places the corner feature using the defined parameters.

❌ Cancel

Clears the selected plate or profile part.

✅ Accept

Accepts the selected plate or profile part.

Shape

Select the shape for the corner feature from the list. The list of available shapes is determined by the type of part that you have selected and the rules that you have defined.

📑 NOTES

- During initial creation of the feature, click **Accept** to show selections in the **Shape** list.

- If the shape is the same for each corner feature, that value displays in the box and a list of other available shapes displays in the list. If you select a different shape, that shape is then applied to all of the selected corner features.

- If the shapes are not the same for all the selected corner features, then the box is blank and the list displays the available shapes. If you select a shape, that shape is then applied to all of the selected corner features. If you do not select a shape, the software creates the corner features according to the rule default values.

- If there are no common shapes, the **Shape** box is disabled and the software creates the corner features according to the rule default values.

Rule Based

Enables or disables the **Shape** list. Selecting this check box disables the **Shape** list.

Place multiple corner features

1. Click **Place Multiple Corner Feature** ⬟.

2. Select the plate systems, plate parts, assemblies, or blocks to which to add the corner features.

3. Click **Accept** ✅.

   📝 **NOTE** The **Rule Based** check box must be clear to select the corner features shapes.

4. If the shape needs to be different than the shape selected by the rule, select the value from the **Shape** list.

5. Click **Finish**.
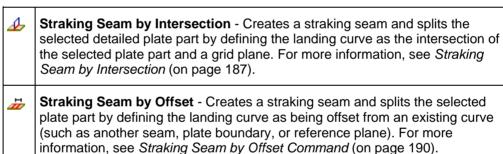
S ECTION 17

Creating Straking Seams

The commands in the straking seams category provide the method to place one or more straking seams in the model. By using straking seams, you can split detailed plate parts into sizes that can be manufactured, as determined by what can be purchased from a mill and what can be built in a work center. Creating a straking seam immediately splits the part and creates the associated assembly and physical connections. Plate parts split by straking seams have the original part of their parent.

📑 NOTES

- Straking seams can only be placed on detail parts. Therefore, you must run **Execute Detailing** 🖋 before you can place straking seams.

- When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where you do not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts. For more information, see *Splitting Objects and Permission Groups* (on page 186).

📐	**Straking Seam by Intersection** - Creates a straking seam and splits the selected detailed plate part by defining the landing curve as the intersection of the selected plate part and a grid plane. For more information, see *Straking Seam by Intersection* (on page 187).
📐	**Straking Seam by Offset** - Creates a straking seam and splits the selected plate part by defining the landing curve as being offset from an existing curve (such as another seam, plate boundary, or reference plane). For more information, see *Straking Seam by Offset Command* (on page 190).
📐	**Straking Seam by Projection** - Creates a straking seam and splits the selected detailed plate part by projecting a 2D landing curve that you have sketched onto the plate part. For more information, see *Straking Seam by Projection Command* (on page 194).
📋	**Straking Seam by Table** - Creates a straking seam and splits the selected detailed plate part by defining the landing curve as coordinates in a table. For more information, see *Straking Seam by Table* (on page 198).

Identifying Seams in Workspace Explorer

You can identify the type of seam in the **Workspace Explorer** by the icon.

- 🔴 Design seams

- 🔵 Planning seams

- ⚏ Straking seams

- ⚏ Intersection seams
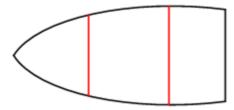
Intended Use of Seams

Intersection Seams

Created in the Molded Forms task at the intersection of two parts, such as where a deck and a bulkhead intersect. Intersection seams are automatically created by the Execute Split command in Molded Forms.
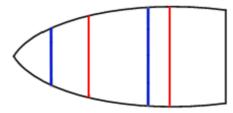
Design Seams

Created in the Molded Forms task using a seam command. A design seam is used where a design property of a system is changing, such as where the thickness of a plate changes. Leaf systems under a single parent system are created.
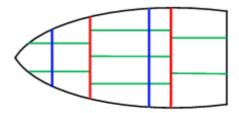
Planning Seams

Create in the Planning task using **Manage Block Intersections** or in the Molded Forms task using a seam command. A planning seam designates the limits of blocks (large segments of a ship assembled in a single location) or smaller assemblies when the limits do not coincide with design seams.

Straking Seams

Created in the Structural Detailing task to designate separate physical plates, usually due to restrictions in plate sizes that can be ordered from the mill. Plate parts under a single leaf system are created.

Splitting Objects and Permission Groups

When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where you do not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts.

📝 **NOTE** If the user who performs the **To Do List** update has write access to both permission groups, you can avoid an iterative update process with multiple users between the two parts.

Example 1

A plate system is defined and the part is detailed. Both the system and the part belong to permission group one. User A has write permission to permission group one, and adds a design seam. The software deletes the original detailed part, creates two new parts, and assigns them to permission group one, because that is the permission group associated with the original part.

Example 2

A plate system is defined in permission group one. The part is moved to permission group two, and detailed. User A has write access to permission group one and read access to permission group two, and adds a design seam. The software places the original detailed part on the **To Do List** in a **To Be Deleted** state. The new parts, along with their assembly connections and physical connections are placed in the active permission group. Any features on the original part migrate to the new parts, and therefore are also assigned to the active permission group.

User B has write access to permission group two, so he can update the **To Do List** which deletes the original part.

User C has write access to both permission group one and two. He can assign the new parts to permission group two. The associated connections and features change to permission group two automatically when the permission group is changed on the part.

Example 3

A plate system is defined in permission group one. The system is split creating parts 1 and 2. Both parts have been assigned to permission group two and detailed. User A, who only has write access to permission group 1, deletes the seam. Parts 1 and 2, and the part level connections between them, go to the **To Do List** in a **To Be Deleted** state. The newly created part 3 is assigned to the active permission group, along with any features that migrated from parts 1 and 2.

Example 4

A bulkhead is bounded by a deck. The bulkhead systems are in permission group PGS1. The deck systems are in permission group PGS2. The bulkhead parts are in permission group PGP1. The deck parts are in permission group PGP2. The deck is split by a seam such that the bulkhead will be bounded by two deck parts. If a split is created by a user that only has access to the deck systems (PGS2), the original leaf logical connection, assembly connection and physical connection between the deck and bulkhead (which are children of the bulkhead and are in PGS1 and PGP1) are placed on the **To Do List** in a **To Be Deleted** state. The two new leaf logical connections, assembly connections, and physical connections are created in the active permission group.

A user with write access to PGS1 must update the **To Do List** to remove the **To Be Deleted** logical connection. A user with write permission to PGP1 must update the **To Do List** to remove the **To Be Deleted** assembly connections and physical connections. A user with write access to both PGS1 and the active permission group (or the permission group where the objects were created if the active permission group has changed) must reassign the new logical connections to the bulkhead permission group (PGS1). A user with write access to PGP1, PGP2, and the active permission group must reassign the new assembly connections and physical connections.

NOTE Use the Database Integrity **Clean** command to correct the permission group assignments, or create a filter to select the necessary objects and edit the permission groups.

See Also

Straking Seam by Intersection

Creates a straking seam on the selected plate system by defining the landing curve of the seam as the intersection of the selected plate system and a grid plane.

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate part boundary, including openings, or another seam on the same plate part.

NOTE When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where you do not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts. For more information, see *Splitting Objects and Permission Groups* (on page 186).

Straking Seam by Intersection Ribbon

Displays the options that you use to place a straking seam in the model.

Seam by Intersection Properties

Activates the **Seam By Intersection Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. For more information, see *Seam By Properties Dialog Box* (on page 190).

Plate Part

Select the plate parts to split. You can select multiple plate parts to place multiple straking seams at the same time. The software uses the same boundaries for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

Intersecting Objects

Select the objects that intersect the plate system for which to generate seams.

Define Boundaries

Define the boundaries of the seam. You can define the boundaries by selecting objects in the model. If you select objects in the model, and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the **Define Boundary** controls described below.

Finish

Creates the seam using the defined parameters.

Cancel

Clears all selected plates or boundaries.

Accept

Accepts all selected plates or boundaries.

Define Boundary Controls

Pick Boundaries

Selects objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 97).

What do you want to do?

- *Place a seam by intersection* (on page)
- *Modify a seam* (on page 189)
- *Delete a seam* (on page 189)

Place a seam by intersection

1. Click **Straking Seam by Intersection** 🦋 on the vertical toolbar.

2. Select the plate system to receive the seams and butts.

3. Click **Accept** ✅.

4. Select an intersecting grid plane or intersecting plate where you want the seam.

5. Click **Accept** ✅.

6. Define the seam boundaries using one or more of the following methods:

    Pick boundaries (on page 34)

    Define a boundary curve (on page 35)

7. Click **Accept** ✅.

8. Click **Finish** to place the seam.

📝 **NOTES**

▪ If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

▪ A straking seam is always attached to a detailed plate part.

▪ You can define properties for the seam to provide a unique identification.

Modify a seam

1. Click **Select** 🦢 on the vertical toolbar.

2. Select **Seams** in the **Locate Filter** box.

3. Select the seam to modify.

4. Use the commands on the ribbon to edit the seam as needed.

📝 **NOTE** When you select multiple objects, you can change the properties that those objects have in common. If the value of the property is the same for the objects that you have selected, that value displays in the ribbon bar. If the values are different, the box for that value is blank. Changing the value of a property makes that the value for all of the selected objects.

Delete a seam

1. Click **Select** 🦢 on the vertical toolbar.

2. Select **Seams and Knuckles** in the **Locate Filter** box.

3. Select the seam to delete.

4. Click **Edit > Delete**.

Seam By Properties Dialog Box

Specifies the properties for the seam that you are placing or editing.

See Also
Main Tab (Seam by Properties Dialog Box) (on page 190)

Main Tab (Seam by Properties Dialog Box)

Specifies the properties for the seam that you are placing or editing.

Main Tab (Seam Properties Dialog Box)

Relationship Tab (on page 400)

Configuration Tab (on page 401)

General Tab

Straking Seam by Offset Command

🖭 Creates a straking seam on the selected plate part by defining the landing curve as being offset from an existing curve (another seam, plate boundary, profile, or grid plane).

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate part boundary, including openings, or another seam on the same plate part.

📝 NOTE When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where you do not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts. For more information, see *Splitting Objects and Permission Groups* (on page 186).

Straking Seam by Offset Ribbon

Displays the options that you use to place an offset seam in the model.

📑 **Seam by Offset Properties**

Activates the **Seam By Offset Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. For more information, see *Seam By Offset Properties Dialog Box* (on page 193).

🔷 **Plate System**

Select the plate parts that you want to split. You can select multiple plate parts to place multiple straking seams at the same time. The software uses the same information for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

〰 **Base Curve**

Select an existing curve or grid plane from which to offset the seam. The intersection of the grid plane and the plate part defines the base curve.

Offset Curve Definition

Define the offset parameters and direction from the base curve. See the **Offset Curve Definition** controls described below.

Boundary Definition

Define the boundaries of the seam. You can define the boundaries by selecting objects in the model. If you select objects in the model, and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the **Boundary Definition** controls described below.

Finish

Places the seam using the defined parameters.

Cancel

Clears all selected objects.

Accept

Accepts all selected objects.

Offset Curve Definition Controls

Unlimited

Defines offset curves without a stop limit.

Stop Limit

Defines offset curves with a stop limit. By using a stop limit, you can select an edge, profile landing curve, connection reference curve, or seam to control the landing curves being created. If you type an offset value and the **Offset** lock is locked, then landing curves are applied at increments of the offset value until the selected stop limit is reached. If the **Offset** lock is unlocked and a value has been defined for **Count**, then the landing curves equal to the number typed in the **Count** box are equally spaced between the base curve and the stop limit.

Offset lock

Locks the **Offset** value, disabling updates of the offset value by mouse moves. This option is only available when you use the **Offset from a Plane** option.

Offset

Specify the offset distance between the base curve and the landing curve that you are defining. If you type 0, the landing curve is place on the base curve. This box may also display the offset distance as defined by mouse movements.

Method

Select an offset method. You can select transverse, longitudinal, vertical, or girth.

Count

Specify how many seams should be created, each equally spaced by increments of the offset value.

Step

Defines an increment that the system should use as you dynamically move the mouse. If

you have typed a value in the **Offset** box, then this control is not used.

Boundary Definition Controls

Pick Boundaries

Selects objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 97).

Place a straking seam by offset

1. Click **Straking Seam by Offset** on the vertical toolbar.

2. Select the plate parts to receive the seam.

3. Click **Accept** .

4. Select a base curve that the software is to use to compute the offset curve.

5. Click **Accept** .

6. Type an offset value in the **Offset** box.

7. Specify the method and count to use.

8. Select, in a graphic view, on which side the base curve to apply the offset. A red arrow appears as you move the mouse indicating the direction of the offset.

9. Click **Accept** .

10. Define the seam boundaries using one or more of the following methods:

    Pick boundaries (on page 34)

    Define a boundary curve (on page 35)

11. Click **Accept** .

12. Click **Finish** to place the seam.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

- A straking seam is always attached to a plate part.

- You can define properties for the seam to provide a unique identification.

Seam By Offset Properties Dialog Box

Specifies the properties for the seam that you are placing or editing.

See Also

Main Tab (Seam By Offset Properties Dialog Box) (on page 193)

Main Tab (Seam By Offset Properties Dialog Box)

Specifies the general properties of the seam.

Name

Specifies the name of the seam.

Rule

Select the naming rule to use to name the seam.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<"IJSeam"><Index Number>. For example, D0-1DCK-1-IJSeam1, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and IJSeam1 is "IJSeam" appended by an index number.

- Select **User Defined** to name the seam using the appropriate box.

Type

Specifies the type of seam that you are placing: **Design Seam** or **Planning Seam**. The list is defined by the StructSeamType codelist.

Landing Curve Definition Method

Displays how the landing curve for the seam was defined.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

See Also

Seam By Offset Properties Dialog Box (on page 193)

Straking Seam by Projection Command

Creates a seam on the selected plate part by projecting a 2D landing curve that you have sketched onto the plate part.

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate boundary, including openings, or another seam on the same plate system.

NOTE When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where you do not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts. For more information, see *Splitting Objects and Permission Groups* (on page 186).

Straking Seam by Projection Ribbon

Displays the options that you use to place a design seam in the model.

Seam by Projection Properties

Activates the **Seam By Projection Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. The initial settings default from the parent plate system. For more information, see *Seam By Projection Properties Dialog Box* (on page 197).

Plate Part

Select the plate part for the seam. You can select multiple plate parts to place multiple seams at the same time. The software uses the same information for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

Landing Curve Definition

Select to draw the landing curve. See the **Landing Curve Definition** controls described below.

Boundary Definition

Define the boundaries of the seam. You can define the boundaries by selecting objects in the model. If you select objects in the model, and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the **Boundary Definition** controls described below.

Finish

Places the seam using the defined parameters.

Cancel

Clears all selected objects.

Accept

Accepts all selected objects.

Landing Curve Definition Controls

Sketching Plane

Specify the sketching plane for the curve. This is the first step in defining the curve.

Add Intersecting Item

Select objects in the model that intersect the sketching plane to use as reference geometry. These extra objects appear in the Draft 2D environment.

Add Projection Item

Select objects in the model that do not intersect the sketching plane to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Draft 2D environment.

Sketch 2D

Opens the Draft 2-D environment in which you can draw the curve.

Auto

Allows the software to automatically add all objects that are relative to the object to be sketched to a select set. Items in the select set are highlighted in the graphics view and in the **Workspace Explorer**. If no objects are added to the select set, the software displays a message in the status bar. This option is only available when you use the **Add Intersecting Item** or the **Add Projection Item** option.

Boundary Definition Controls

Pick Boundaries

Selects objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 97).

Place a straking seam by projection

1. Click **Straking Seam by Projection** on the vertical toolbar.

2. Select the plate part for the straking seam.

3. Click **Accept** .

4. Specify the plane for the seam sketch using one of the following methods:

   Define plane using angle from plane (on page 29)

   Define a coincident plane (on page 29)

   Define an offset plane (on page 30)

   Define a plane using three points (on page 31)

   Define a plane using a point and a normal vector (on page 32)

5. Click **Add Intersecting Item** .

6. Select objects in the Model that intersect the sketching plane that you want to use as reference geometry. These extra objects will appear in the Sketch 2D environment.

7. Click **Add Projection Item** .

8. Select objects in the Model that do not intersect the sketching plane to use as reference geometry. These extra objects are projected onto the sketching plane and appear in the Sketch 2D environment.

9. Click **Sketch 2D** .

10. In the Sketch 2D environment, sketch the seam curve.

11. Click **Finish** in the Sketch 2D environment.

12. Click **Accept** .

13. Define the seam boundaries using one or more of the following methods:

    Pick boundaries (on page 34)

    Define a boundary curve (on page 35)

14. Click **Accept** .

15. Click **Finish** to place the seam.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

- A straking seam is always attached to a plate part.

- You can define properties for the seam to provide a unique identification.

Seam By Projection Properties Dialog Box

Specifies the properties for the seam that you are placing or editing.

See Also
Main Tab (Seam By Projection Properties Dialog Box) (on page 197)

Main Tab (Seam By Projection Properties Dialog Box)

Specifies the general properties of the seam.

Name

Specifies the name of the seam.

Rule

Select the naming rule to use to name the seam.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<"IJSeam"><Index Number>. For example, D0-1DCK-1-IJSeam1, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and IJSeam1 is "IJSeam" appended by an index number.

- Select **User Defined** to name the seam using the appropriate box.

Type

Specifies the type of seam that you are placing: **Design Seam** or **Planning Seam**. The list is defined by the StructSeamType codelist.

Landing Curve Definition Method

Displays how the landing curve for the seam was defined.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

See Also
Seam By Projection Properties Dialog Box (on page 197)

Straking Seam by Table

Creates a straking seam on the selected plate part by defining the landing curve as coordinates in a table.

You can control the boundaries of the seam by selecting the parent plate system, seam, or split knuckle line that intersects the landing curve. If you do not define boundaries, the software uses the landing curve ends as the seam boundaries. Seams must end at a plate part boundary, including openings, or another seam on the same plate system.

NOTE When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where you do not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts. For more information, see *Splitting Objects and Permission Groups* (on page 186).

Straking Seam By Table Ribbon

Displays the options that you use to place a straking seam in the model.

Seam by Table Properties

Activates the **Seam By Table Properties** dialog box, which you use to view and modify the properties of the seam that you are about to place in the model. For more information, see *Seam By Table Properties Dialog Box* (on page 204).

Plate Part

Select the plate part for the seam. You can select multiple plate parts to place multiple seams at the same time. The software uses the same information for each seam. After placement, the seams are unrelated to each other and can be edited as individual entities.

Landing Curve Table

Activates a table where you can define the landing curve coordinates for the seam. For more information, see *Landing Curve Table Dialog Box* (on page 201).

Boundary Definition

Define the boundaries of the seam. You can define the boundaries by selecting objects in the model. If you select objects in the model, and those objects are moved, the software automatically resizes the seam to maintain the boundary relationship. See the Boundary Definition controls described below.

Finish

Places the seam using the defined parameters.

Cancel

Clears all selected objects.

Accept

Accepts all selected objects.

Boundary Definition Controls

Pick Boundaries

Selects objects in the model or in the **Workspace Explorer** to define the boundaries. This option is only available when you are defining the boundaries.

Solve Ambiguity

If the objects that you selected in the model for the boundary define an ambiguous solution, then the software prompts you to select one bounded section to clarify the needed boundary. This ambiguity only occurs if one of the boundaries wraps around both sides of the profile or seam, such as the hull. Move the cursor over a bounded section, and then click to select.

Boundary List

Opens the **Boundary List** dialog box. This dialog box allows you to review and define the boundaries. For more information, see *Boundary List Dialog Box* (on page 97).

What do you want to do?

- *Place a straking seam by table* (on page 199)
- *Set coordinate by plane* (on page 200)
- *Set coordinate by key-ins* (on page 200)
- *Derive points* (on page 200)
- *Export points to Microsoft Excel* (on page 201)
- *Import points from Microsoft Excel* (on page 201)

Place a straking seam by table

1. Click **Straking Seam by Table** on the vertical toolbar.
2. Select the plate part for the seam.
3. Click **Accept** .
4. Define the landing curve coordinates in the table. For more information about defining coordinates, see *Landing Curve Table Dialog Box* (on page 201).
5. Click **Close**.
6. Click **Boundaries** to apply boundaries.
7. Define the seam boundaries using one or more of the following methods:

    Pick boundaries (on page 34)

    Define a boundary curve (on page 35)
8. Click **Accept** .

9. Click **Finish** to place the seam.

NOTES

- If you do not select boundaries, the software uses the entire length of the base curve to apply to the resulting seam curves.

- A seam is always associated with a plate part.

- You can define properties for the seam to provide a unique identification.

Set coordinate by plane

1. Click in the **Reference** cell for the coordinate.

2. Click **Browse** .

3. Select **Reference Plane**.

4. Select a coordinate system plane to use as the coordinate.

5. Click **OK**.

6. Type an offset for the coordinate from the plane, if needed, in the **Offset** box.

Set coordinate by key-ins

1. Click in the **Reference** cell for the coordinate.

2. Click **Browse** .

3. Select **Coordinate**.

4. Type a coordinate location in the box. You must include a unit of measure, such as **m** for meter.

5. Click **OK**.

Derive points

1. In row 1, define one end point of the landing curve. Be sure to select the **Definition Point** option.

2. In row 2, define the other end point of the landing curve. Be sure to select the **Definition Point** option.

3. Select the axes to derive points for. In general, you want to select an axis that is perpendicular to the direction of your landing curve.

4. Select the coordinate system or object in the model along which to derive the points.

5. Click **Preview**.

6. Review the points in the table that the software derived.

NOTES

- You can change a derived point to be a definition point by selecting the **Definition Point** option.

- Click **Preview** when you make any changes to the definition points in the table.

Export points to Microsoft Excel

1. Define points in the table.

2. Highlight the cells to export.

   TIP You can highlight the cells by dragging the pointer over them, or you can click the blank cell in the upper left corner to select all of the points in the table.

3. Right-click in the table, and select **Copy Cell**.

4. Open a Microsoft Excel workbook.

5. Select cell A1 on a workbook sheet.

6. Click **Edit > Paste** in Microsoft Excel.

7. Save the Microsoft Excel workbook.

Import points from Microsoft Excel

1. Open the **Landing Curve Table** dialog box.

2. Click **Import**.

3. Select the Microsoft Excel file that contains the points to import.

4. Click **OK**.

Landing Curve Table Dialog Box

Specifies a landing curve by defining X-, Y-, and Z-coordinates of points along the landing curve. Each point along the landing curve is defined on a single row in the table. You must define two out of the three point coordinates. The software derives the third coordinate for you, and highlights the derived coordinate in blue.

	X		Y		Z		Definition Point	KNU
	Reference	Offset	Reference	Offset	Reference	Offset		
1	F 10.000 m	0.00 m	7.85 m	0.00 m	D 11.000 m	0.00 m	☑	☐
2	10.27 m	0.00 m	8.00 m	0.00 m	10.98 m	0.00 m	☐	☐
3	12.11 m	0.00 m	9.00 m	0.00 m	10.88 m	0.00 m	☐	☐
4	13.95 m	0.00 m	10.00 m	0.00 m	10.80 m	0.00 m	☐	☐

X Reference

Select a transverse bulkhead, frame plane, or another YZ grid plane to specify the X-coordinate of the landing curve point. You can also explicitly specify the X-coordinate by typing the point's location along the X-axis.

X Offset

Type an offset value from the value in the **X Reference** box.

Y Reference

Select a buttock plane, longitudinal plane, or another XZ grid plane to specify the Y-coordinate of the landing curve point. You can also explicitly specify the Y- coordinate by typing the point's location along the Y-axis.

Y Offset

Type an offset value from the value in the **Y Reference** box.

Z Reference

Select the waterline plane, horizontal plane, or another XY grid plane to specify the Z-coordinate of the landing curve point. You can also explicitly specify the Z- coordinate by typing the point's location along the Z-axis.

Z Offset

Type an offset value from the value in the **Z Reference** box.

Definition Point

Select this option if the coordinate on the row defines a point on the landing curve.

TIP Points for which you have typed coordinates are usually definition points. Coordinates that are not definition points are not saved as part of the landing curve definition.

KNU

Select this option if the coordinate on the row defines a knuckle point.

Curvature Vector Display Method

Specifies how the curvature vectors appear when you click **Preview**. Select **Number of vectors** to specify a fixed number of vectors that the software is to distribute evenly along the landing curve. Specify the number of vectors in the **Count** box. Select **Vector spacing** to place vectors along the landing curve a fixed distance apart. The software determines the number of vectors based on the landing curve length and the spacing distance specified in the **Spacing** box.

Count

Specifies the number of vectors to distribute evenly along the landing curve preview. This option is only available when **Curvature Vector Display Method** is set to **Number of vectors**.

Spacing

Specifies the distance between vectors along the landing curve preview. This option is only available when **Curvature Vector Display Method** is set to **Vector Spacing**.

Range

Select where along the landing curve preview to display curvature vectors. Select **Entire Curve** to display curvature vectors along the entire landing curve. Select **Between Definition Points** to display curvature vectors only between definition points.

Scaling Factor

Define a scaling factor value for the curvature vectors. The larger the value, the larger the curvature vectors appear in the graphic view.

Derived Points in the Table
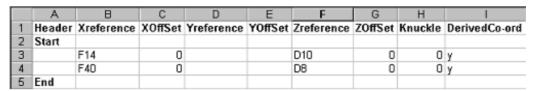
These options control when and how the software derives points between definition points that you define.

- **X: Transverse** - Select this option to have the software derive points for each X grid plane that intersects the landing curve. You must select a coordinate system.

- **Y: Buttock** - Select this option to have the software derive points for each Y grid plane that intersects the landing curve. You must select a coordinate system.
- **Z: Waterline** - Select this option to have the software derive points for each Z plane that intersects the landing curve. You must select a coordinate system.

Import From File

Reads landing curve points into the table from a Microsoft Excel workbook. The workbook must contain one sheet and the sheet must be formatted as shown. Defined references must be typed exactly in the model, as shown for **Xreference** and **Zreference**. Derived references are left blank, as shown for **Yreference**.

	A	B	C	D	E	F	G	H	I
1	Header	Xreference	XOffSet	Yreference	YOffSet	Zreference	ZOffSet	Knuckle	DerivedCo-ord
2	Start								
3		F14	0			D10	0	0	y
4		F40	0			D8	0	0	y
5	End								

Preview

Displays the landing curve, as currently defined by the points in the table, in the model. You can select **Preview** multiple times to see changes made to the landing curve as you add and edit points in the table.

Print

Prints a copy of the landing curve table as it is currently defined.

Close

Closes the dialog box.

Right-Click Menu Commands

There are several right-click commands available to help in defining the points in the table.

Cut Cell

Removes the contents of the selected cell and moves the contents to the Clipboard.

Copy Cell

Moves a copy of the selected cell contents to the Clipboard.

Paste Cell

Places a copy of the Clipboard contents in the cell.

Cut Row

Removes the contents of the selected row and moves the contents to the Clipboard.

Copy Row

Moves a copy of the selected row contents to the Clipboard.

Paste Row

Places a copy of the Clipboard contents in the row, replacing any existing values in the row.

Paste Reverse

If more than one row is copied, this command places a copy of the Clipboard contents in the

same number of rows but in reverse order, replacing any existing values in the row.

Insert Cut/Copied Rows

Inserts the row on the Clipboard into a new row in the table.

Insert Reverse Rows

If more than one row is copied, this command inserts the rows from the Clipboard in the table in reverse order.

Delete

Removes the contents of the selected cell.

Insert Row

Inserts a new, empty row into the table above the selected row.

Remove Row

Removes the selected row from the table.

Make Derived

Marks the selected reference as the one that the software derives.

Seam By Table Properties Dialog Box

Specifies the properties for the seam that you are placing or editing.

See Also
Landing Curve Table Dialog Box (on page 201)
Main Tab (Seam By Table Properties Dialog Box) (on page 204)

Main Tab (Seam By Table Properties Dialog Box)

Specifies the general properties of the seam.

Name

Specifies the name of the seam.

Rule

Select the naming rule to use to name the seam.

- Select **StdHierarchyChildNamingRule** to use the following syntax: <Parent Name>-<"IJSeam"><Index Number>. For example, D0-1DCK-1-IJSeam1, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and IJSeam1 is "IJSeam" appended by an index number.

- Select **User Defined** to name the seam using the appropriate box.

Type

Specifies the type of seam that you are placing: **Design Seam** or **Planning Seam**. The list is defined by the StructSeamType codelist.

Landing Curve Definition Method

Displays how the landing curve for the seam was defined.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

See Also

Seam By Table Properties Dialog Box (on page 204)

Modify Profile Straking Seam

Select an existing straking seam to modify its properties.

Modify Profile Straking Seam Ribbon

Displays the options that you use to modify a straking seam.

Properties

Activates the **Profile Straking Seam Properties** dialog box, which you can use to edit properties for the seam. For more information, see *Profile Straking Seam Properties Dialog Box* (on page 206).

Profile System

Select the profile system or the plate system that has associated profile systems upon which to place a straking seam. The seams are placed on the child detailed parts of the profile systems. You can select stiffener profile systems or linear beam profile systems.

References

Select objects that intersect the detailed profile part at which to place the seam. You can select objects in either the graphic view or in the **Workspace Explorer**. Objects that you can select as references include: reference curves, reference planes, plate systems, edge reinforcement systems, seams (design, planning, and straking), profile systems (landing curve), and edge ports including openings.

Finish

Modifies the seam using the parameters that you have defined.

Cancel

Rejects the selected objects.

Accept

Accepts the selected objects and advances the command to the next step.

Offset Method

Select the offset method, if needed.

Offset Value

Type the offset value. You can specify the direction of the offset by typing a plus (+) or minus (-) sign in front of the offset value. A positive value is offset in the positive direction of the closest main coordinate system axis.

Profile Straking Seam Properties Dialog Box

Specifies the properties for the profile straking seam.

See Also
Main Tab (Profile Straking Seam Properties Dialog Box) (on page 206)

Main Tab (Profile Straking Seam Properties Dialog Box)

Specifies the general settings for the profile straking seam.

Name

Displays the name to assign to the profile straking seam. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Specifies how the name is created. If set to a naming rule, the software automatically generates the name for you. If you select **User Defined**, you can type in the straking seam name in the **Name** box.

Type

Displays the object type, which in this case is straking seam.

Parent System

Displays the name of the parent system of the straking seam.

See Also
Profile Straking Seam Properties Dialog Box (on page 206)

S E C T I O N   1 8

Split Physical Connection

Splits an existing physical connection at a location that you specify.

NOTE When an object is split, or a split is deleted, the software places the resulting parts in the same permission group as the original parts. In ambiguous cases where the user does not have permission to update parts or connections, the software places the original parts on the **To Do List** in a **To Be Deleted** state and assigns the new parts to the active permission group. A user with write access to the permission group assigned to the original parts must update the **To Do List** to delete the original parts. For more information, see *Splitting Objects and Permission Groups* (on page 186).

Split Physical Connection Ribbon

Displays the options that you use to split an existing physical connection at a location that you specify.

Split Point Properties

Activates the **Split Point Properties** dialog box. This option is only available in the modification mode. For more information, see *Split Point Properties Dialog Box* (on page 209).

Physical Connection

Select one or more physical connections to split, either graphically in the active window or through the **Workspace Explorer**.

Reference

Select a reference for positioning the split locations.

Split Point Definition by Geometric Construction

Allows creation of the split point directly in a graphic view by using the **Geometric Construction Palette** dialog box, the **Geometric Construction** ribbon, and the **Geometric Construction Explorer**. This option is the 3D equivalent to Sketch 2D. For more information, see Geometric Construction Palette Dialog Box.

Finish

Processes the **Split Physical Connection** command, and commits the changes to the database.

Reference Definition Method

Defines how the split point is created. Click **Reference Definition Method** to create a reference to the split point using a point on the physical connection. This point is used as a reference. If it is not selected, the split point is created with the selected object as the reference. **Reference Definition Method** is not selected by default. Press SHIFT to toggle between using the points or the objects as a reference.

❌ **Cancel**

> Cancels the selection.

✅ **Accept**

> Accepts the selection.

Offset Method

> Select the offset method, if needed.

Split a physical connection

Split a physical connection by point

1. Click **Split Physical Connection** .

2. Select one or more physical connections to split, either graphically in the active window or through the **Workspace Explore**.

3. Click **Accept** ✅.

4. Click **Reference** .

5. Click **Reference Definition Method** .

6. Select a point on the physical connection geometry to use as a reference for positioning the split location.

7. Click **Finish** to commit the split to the database.

Split a physical connection by object

1. Click **Split Physical Connection** .

2. Select one or more physical connections to split, either graphically in the active window or through the **Workspace Explore**.

3. Click **Accept** ✅.

4. Click **Reference** .

5. Select a reference to position the split location.

6. Select **Offset Method** and **Distance**, if required.

7. Click **Finish** to commit the split to the database.

Split a physical connection by geometry construction

1. Click **Split Physical Connection** .

2. Select one or more physical connections to split, either graphically in the active window or through the **Workspace Explore**.

3. Click **Accept** ✅.

4. Click **3D** .

*The **Geometric Construction Palette** dialog box displays. The **Geometric Construction** ribbon and the **Geometric Construction Explorer** also display.*

5. Create a geometric construction point at which to position the split location.

6. Click **Finish** to commit the split to the database.

▤ NOTES

- The split physical connections are also displayed below the split point in the **Workspace Explorer** under the parent physical connection.

- You can also split a physical connection where the reference crosses through a gap by selecting the required offset method and value.

- You can also press CTRL + ENTER to commit the split to the database.

- **Offset Method** is only available when you split a physical connection by object.

- If you select multiple physical connections by point or by geometry construction, the software splits the physical connection at only one point.

Split Point Properties Dialog Box

Specifies the properties for the split point that you are editing.

See Also
General Tab (Split Point Properties Dialog Box) (on page 209)
Relationship Tab (on page 400)
Configuration Tab (on page 401)

General Tab (Split Point Properties Dialog Box)

Specifies the properties for the split point.

Category

Select the type of properties to view for the selected split point.

Name

Specifies the name of the split point.

Name Rule

Select the name rule to use to name the split point.

▤ NOTE The format for a default name rule is <parent physical connection name>-split point-<naming counter>. If you change the name of the split point, the name rule changes to **User Defined**.

Geometric Construction Palette Dialog Box

Specifies a geometric construction point interface. The interface allows you to create point geometry directly in a graphic view. The **Geometric Construction Palette** appears when you click the 🔧 option on the **Split Physical Connection** 🔨 ribbon .

The **Geometric Construction Explorer** also appears when 🔧 is clicked. As geometry is created, a hierarchical list of applicable surfaces, curves, and points display.

Filters

> Limits the available macros on a tab to the selected type.

Clear Recent

> Removes recently selected macros from the **Recent** tab.

? Help
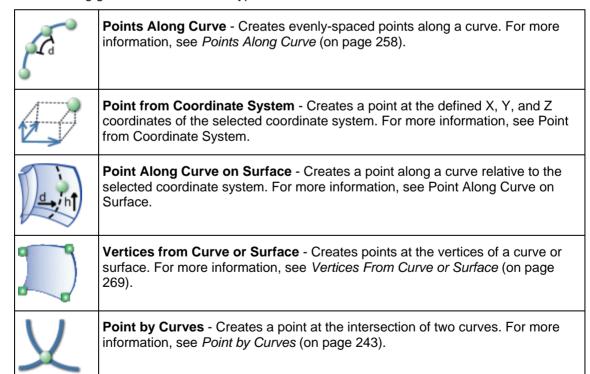
> Displays help for the selected macro in a preview window. If no help is available, a preview of the selected macro displays.
>
> 📄 **NOTE** You can also select **Tools** > **Automatic Preview** to display the preview window.

The **Geometric Construction Palette** contains the following tabs:

- **Points** - Displays interfaces for creating points.

- **Recent** - Displays geometric construction interfaces recently selected on the other tabs. Click **Clear Recent** to remove interfaces from this tab.

The following geometric construction types are available.

	Points Along Curve - Creates evenly-spaced points along a curve. For more information, see *Points Along Curve* (on page 258).
	Point from Coordinate System - Creates a point at the defined X, Y, and Z coordinates of the selected coordinate system. For more information, see Point from Coordinate System.
	Point Along Curve on Surface - Creates a point along a curve relative to the selected coordinate system. For more information, see Point Along Curve on Surface.
	Vertices from Curve or Surface - Creates points at the vertices of a curve or surface. For more information, see *Vertices From Curve or Surface* (on page 269).
	Point by Curves - Creates a point at the intersection of two curves. For more information, see *Point by Curves* (on page 243).

	Point by Curve and Surface - Creates a point at the intersection of a curve and a surface. For more information, see *Point by Curve and Surface* (on page 239).
	Point by Surfaces - Creates a point at the intersection of three surfaces. For more information, see *Point by Surfaces* (on page 246).
	Point Along Curve - Creates a point at a distance along a curve. For more information, see *Point Along Curve* (on page 211).
	Point at Curve Middle - Creates a point at the middle of a curve. For more information, see *Point at Curve Middle* (on page 231).
	Point at Curve Start - Creates a point at the beginning of a curve. For more information, see *Point at Curve Start* (on page 236).
	Point at Curve End - Creates a point at the end of a curve. For more information, see *Point at Curve End* (on page 228).

NOTE After you select an interface, the ribbon for that interface appears.

The **Geometric Construction Explorer** also appears when is clicked. As geometry is created, a hierarchical list of applicable surfaces, curves, and points display.

Geometric Construction Explorer **(on page 273)**

Point Along Curve

Tooltip: PointAlongCurve

Creates a point at a distance along a curve.

Required Input	Output
Curve, line, or edge	Point
Optional Input	
Reference point	
Projection surface	
Parameters	
Distance	

Point Along Curve Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click ⚒ again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the curve on which the point is placed.

(2) - Specifies a reference point for measurement of the point location. If no reference point is selected, then the starting end of the curve is used. This value is optional.

(3) - Specifies a tracking point used to control the direction of point placement when an ambiguity exists. A tracking point is only needed with some curves, such as:

- A circle defined by two points and a mid-point in which there are two arcs between the start and end points.

- A curve on multiple sides of a non-planar surface, such as a seam or reference curve wrapping around a hull.

This value is optional.

📝 NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

(4) - Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- ▪ 1 - No background color when there is no input.

- ▪ 1 - Yellow background when an input is selected.

- ▪ 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

▤ **NOTE** Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- ▪ 1 and 0.00 m ▼ - No background color when you keep the value of the original geometric construction.

- ▪ 1 and 0.10 m ▼ - Yellow background when you change a value.

- ▪ 1 and 0.10 m ▼ - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

   Clears the selections for the current step.

✅ **Accept**

   Accepts the current selections and displays a preview.

Continue

   Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ **IMPORTANT** The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Distance

Defines the distance of the point from the starting end of the curve, or from optional reference point (2).

Track Flag

Controls how the tracking point (3) is used when an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?

- *Place a point along a curve* (on page)
- *Modify a geometric construction* (on page 267)
- *Change a geometric construction to a different definition* (on page 268)
- *Delete a geometric construction* (on page 268)

Place a point along a curve

1. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

2. If using a reference point, create the point with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

3. Click **PointAlongCurve** on the **Geometric Construction Palette**.

4. Select the same curve.

5. Optionally, click (2), and select the reference point.

6. In the **Distance** box, type a value.

   A preview of the point appears at the defined distance from the start of the curve or the reference point.

7. If more than one solution is possible, click (3), and select the needed value for **Track Flag**.

   NOTE If changing the value for **Track Flag** does not give the needed solution, you may need to first create a point that you then select after clicking (3). The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the Common User's Guide

   Topological Points

   Points (Geometric Construction Palette Dialog Box)

8. Click **Continue**.

> *Temporary geometry is displayed in the graphic view and as **PointAlongCurve** in the **Geometric Construction Explorer**.*

9. If needed, create additional geometric constructions.

10. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point Along Curve on Surface

Tooltip: PointAlongCurveOnSurface

Creates a point along a curve on a surface relative to the selected coordinate system.

Required Input	Output
1 - Coordinate system	Point along the curve
2 -Surface	
3 - Reference plane intersecting the surface	
6 - Supporting plane intersecting the surface	
Optional Input	
Points or curves to define reference plane **3**	
Parameters	
Offset method and value	
Distance method and value	

Point Along Curve on Surface Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric**

Construction dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system. The coordinate system is used when there is an ambiguous solution.

2

Specifies the surface. By default, this is the surface of the plate selected for the main command.

3

Specifies a reference plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the reference curve from which the new point is offset as defined by **Offset** and **Offset Direction**.

(4)

Specifies a point used to define the plane intersecting the surface. This option is only available if a point or curve is selected for **3**.

(5)

Specifies a third point used to define the plane intersecting the surface. This option is only available if points are selected for **3** and **(4)**.

6

Specifies a supporting plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the supporting curve along which the new point is located as defined by **Distance** and **Distance Method**.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- **1** - No background color when there is no input.

- **1** - Yellow background when an input is selected.

- **1** - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

📄 NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- ▪ [1] and [0.00 m ▼] - No background color when you keep the value of the original geometric construction.

- ▪ [1] and [0.10 m ▼] - Yellow background when you change a value.

- ▪ [1] and [0.10 m ▼] - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

Clears the selections for the current step.

✅ **Accept**

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

⭐ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Offset

Defines the offset distance of the point along the reference curve.
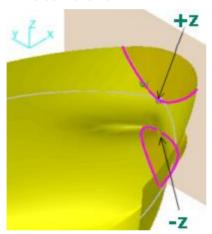
Offset Direction

Defines the offset direction (with respect to the selected coordinate system **1**) along the reference curve. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available when **Offset** is set to a value other than **0**.

Inter Ambiguity

Specifies the solution when the supporting plane intersects a complex surface in multiple locations. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The following example shows the ambiguity options in the bow of a hull.

Distance

Defines the distance of the point from the reference curve along the supporting curve.

Distance Method

Defines how the point is measured along the supporting curve. Select **Girth**, **Along x**, **Along y**, or **Along z**.

Point Ambiguity

Specifies the solution along the supporting curve when **Distance** is not **0** and **Girth** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Girth**.

Distance Direction

Specifies the solution along the supporting curve when **Distance** is not **0** and **Along x**, **Along y**, or **Along z** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Along x**, **Along y**, or **Along z**.

Side Ambiguity

Specifies the solution when the reference plane intersects a complex surface in more than one location, such as on the port and starboard sides of a hull. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**.

⊘ **WARNING** For options that have coordinate axis values, select a value that is compatible with the orientation of the curve. For example, if a non-linear reference curve is in the x-y plane, valid options for **Offset Direction** are **+x**, **-x**, **+y**, and **-y**.

Point Along Curve on Surface Ribbon (for topological points in the Molded Forms task)

📋 Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

Specifies the coordinate system. The coordinate system is used when there is an ambiguous solution.

2

Specifies the surface. By default, this is the surface of the plate selected for the main command.

3

Specifies a reference plane intersecting the surface, or a curve or point used to define the plane. The curve created by this intersection is the reference curve from which the new point is offset as defined by **Offset** and **Offset Direction**.

(4)

Specifies a point used to define the plane intersecting the surface. This option is only available if a point or curve is selected for **3**.

(5)

Specifies a third point used to define the plane intersecting the surface. This option is only available if points are selected for **3** and **(4)**.

6

Specifies a supporting plane intersecting the surface or a curve or point used to define the plane. The curve created by this intersection is the supporting curve along which the new point is located as defined by **Distance** and **Distance Method**.

Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

- **1** - No background when no input is selected.

- **1** - Yellow background when an input is selected.

- **1** - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- [1] and [0.00 m ▼] - No background when the value of the original geometric construction is retained.

- [1] and [0.10 m ▼] - Yellow background when you change a value.

- [1] and [0.10 m ▼] - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- [1] - Blue background when the input is the same for all selected geometric constructions.

- [1] - No background with pink underline when the input is different for *all* selected geometric constructions.

- [1] - Blue background with pink underline when the input is different for *some* selected geometric constructions.

- [1] - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.

- [▼] - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.

Y

[▼]
2.00 m [from GC0SRect400x300x100-1-0101, GC0SRect400x300x100-1-0102, GC0SOval500...
3.00 m [from GC0SCircle300-1-0001, GC0SCircle300-1-0002, GC0SCircle300-1-0003, G...
Shared...

Reject

   Clears the current selection.

Accept

   Accepts the current selections, and displays a preview.

Finish

   Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

   Specifies the name of the geometric construction.

Parent

   Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.

- **More** - Select a system in the model.

NOTE The **New GC Set** option is not used for topological points.

Offset

Defines the offset distance of the point along the reference curve.

Offset Direction

Defines the offset direction (with respect to the selected coordinate system **1**) along the reference curve. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available when **Offset** is not set to **0**.

Inter Ambiguity

Specifies the solution when the supporting plane intersects a complex surface in multiple locations. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. The following example shows the ambiguity options in the bow of a hull.
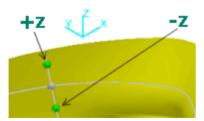

Distance

Defines the distance of the point from the reference curve along the supporting curve.

Distance Method

Defines how the point is measured along the supporting curve. Select **Girth**, **Along x**, **Along y**, or **Along z**.

Point Ambiguity

Specifies the solution along the supporting curve when **Distance** is not **0** and **Girth** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Girth**.
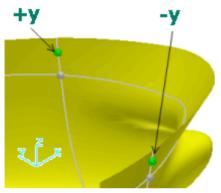

Distance Direction

Specifies the solution along the supporting curve when **Distance** is not **0** and **Along x**, **Along y**, or **Along z** is selected as **Distance Method**. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**. This option is available for **Along x**, **Along y**, or **Along z**.

Side Ambiguity

Specifies the solution when the reference plane intersects a complex surface in more than one location, such as on the port and starboard sides of a hull. Select **+x**, **-x**, **+y**, **-y**, **+z**, or **-z**.

★ IMPORTANT For options that have coordinate axis values, select a value that is compatible with the orientation of the curve. For example, if a non-linear reference curve is in the x-y plane, valid options for **Offset Direction** are **+x**, **-x**, **+y**, and **-y**.

What do you want to do?

- *Place a point on a surface and along a curve* (on page)

- *Modify a geometric construction* (on page)

- *Change a geometric construction to a different definition* (on page)

- *Delete a geometric construction* (on page)

- *Place a topological point on a surface and along a curve* (on page) (in the Molded Forms task)

- *Modify a topological point* (on page) (in the Molded Forms task)

- *Change a topological point to a different point definition* (on page 227) (in the Molded Forms task)

- *Delete a topological point* (on page 228) (in the Molded Forms task)

Place a point on a surface and along a curve
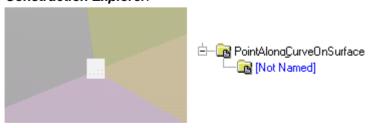
1. Click **PointAlongCurveOnSurface** on the **Geometric Construction Palette**.

2. Select a coordinate system.

3. Select a surface, such as a plate system or reference plane.

   NOTE If a surface was selected for the main command, then this is the default value and **2** has a yellow background. If you want to use this surface, skip this step and click **3**.

4. Select a plane intersecting the surface. The **Offset** direction is along the reference curve created by this intersection.

5. Select another plane intersecting the surface and the reference curve. The **Distance** direction is along the supporting curve created at the intersection with the surface.

6. Select a value for **Distance Method**.

7. Type the needed values for **Offset** and **Distance**.

   A preview point in green appears at the intersection of the surface, reference curve, and supporting curve. If more than one point appears, then more than one solution is possible.

8. If multiple points appear, perform one of the following steps to resolve the ambiguity:

   - If the surface is complex (a plane intersects it more than once), then select a coordinate axis direction for **Inter Ambiguity**.

   - If **Girth** is selected for **Distance Method**, then select a coordinate axis direction for **Point Ambiguity**.

   - If two solutions are possible on opposing sides of the reference curve, then select a coordinate axis direction for **Side Ambiguity**.

   NOTE For each ambiguity, select a value that is compatible with the orientation of the points.

   A single green preview point is now visible.

9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **[Not Named]** in the **Geometric Construction Explorer**.*

10. If needed, create additional geometric constructions.

11. Click **Close** to return to the main command ribbon.

★ **IMPORTANT**

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place a topological point on a surface and along a curve

1. In the Molded Forms task, select **Insert** > **Topological Points**.

2. Click **PointAlongCurveOnSurface** on the **Point Palette**.

3. Select a coordinate system.

4. Select a surface, such as a plate system or reference plane.

5. Select a plane intersecting the surface. The **Offset** direction is along the reference curve created by this intersection.

6. Select another plane intersecting the surface and the reference curve. The **Distance** direction is along the supporting curve created at the intersection with the surface.

7. Select a value for **Distance Method**.

8. Type the needed values for **Offset** and **Distance**.

   A preview point in green appears at the intersection of the surface, reference curve, and supporting curve. If more than one point appears, then more than one solution is possible.
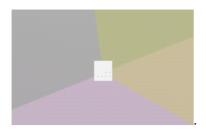
9. If multiple points appear, perform one of the following steps to resolve the ambiguity:

   - If the surface is complex (a plane intersects it more than once), then select a coordinate axis direction for **Inter Ambiguity**.

   - If **Girth** is selected for **Distance Method**, then select a coordinate axis direction for **Point Ambiguity**.

   - If two solutions are possible on opposing sides of the reference curve, then select a coordinate axis direction for **Side Ambiguity**.

   📝 **NOTE** For each ambiguity, select a value that is compatible with the orientation of the points.

   A single green preview point is now visible.

10. Optionally, type a name for the point.

11. Optionally, select **More** in the **Parent** box, and select a parent system for the point. For example, you can select the plate system on which you are placing the point.

12. Click **Finish**.

    *Point geometry is displayed in the graphic view and in the **Workspace Explorer**.*

.

Modify a topological point

1.   On the vertical toolbar, click **Select** .

2.   In a graphic view or in the Workspace Explorer, select the topological point to modify.

The point is highlighted in the graphic view, the ribbon for the topological point appears, and inputs are highlighted in the Workspace Explorer.

3.   Make the needed changes.

4.   Click **Accept** .

A preview of the changed geometric construction appears.

5.   Click **Finish**.

The modified point is displayed in the graphic view and in the Workspace Explorer.

Change a topological point to a different point definition

1.   On the vertical toolbar, click **Select** .

2.   In a graphic view or in the Workspace Explorer, select the topological point to modify.

*The point is highlighted in the graphic view, the ribbon for the topological point appears, and the inputs are highlighted in the **Workspace Explorer**.*

3.   In the **Macro Definition** box, click **More**.

*The **Select Geometric Construction** dialog box appears. Only the appropriate point geometric constructions appear.*

4.   Select the needed point geometric construction, and then click **OK**.

*The new selection appears in the **Macro Definition** box with a yellow background. Options with blue backgrounds contain new values suggested by the software. Options with white backgrounds retain the values from the previous geometric construction.*

5.   Make the needed changes to options on the ribbon.

Changed options have a yellow background.

6.   Click **Accept** .

A preview of the changed point appears.

7.   Click **Finish**.

*The modified point is displayed in the graphic view and in the **Workspace Explorer**.*

Delete a topological point

1. On the vertical toolbar, click **Select** .

2. In a graphic view or in the Workspace Explorer, select the topological point to delete.

   The point is highlighted in the graphic view, the ribbon for the topological point appears, and inputs are highlighted in the Workspace Explorer.

3. Click **Delete** ✕ .

   The geometric construction is deleted.

Point at Curve End

Tooltip: PointAtCurveEnd

Creates a point at the end of a curve.

Required Input	Output
Curve, line, or edge	Point
Optional Input	
Projection surface	

Point at Curve End Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

✕ **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 🔲 again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ **IMPORTANT** Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the curve on which the point is placed.

(2)

Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.

- 1 - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

📄 **NOTE** Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.

- 1 and 0.10 m - Yellow background when you change a value.

- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

Clears the selections for the current step.

✅ **Accept**

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a point at the end of a curve* (on page)

- *Modify a geometric construction* (on page 267)

- *Change a geometric construction to a different definition* (on page 268)

- *Delete a geometric construction* (on page 268)

Place a point at the end of a curve

1. Click **PointAtCurveEnd** on the **Geometric Construction Palette**.

2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

   A preview of the point appears at the end of the curve.

3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointAtCurveEnd** in the **Geometric Construction Explorer**.*

4. If needed, create additional geometric constructions.

5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point at Curve Middle

Tooltip: PointAtCurveMiddle

Creates a point at the middle of a curve.

Required Input	Output
Curve, line, or edge	Point
Optional Input	
Projection surface	

Point at Curve Middle Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the curve on which the point is placed.

(2)

Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- ▪ [1] - No background color when there is no input.

- ▪ [1] - Yellow background when an input is selected.

- ▪ [1] - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

📝 **NOTE** Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- ▪ [1] and [0.00 m ▼] - No background color when you keep the value of the original geometric construction.

- ▪ [1] and [0.10 m ▼] - Yellow background when you change a value.

- ▪ [1] and [0.10 m ▼] - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

Clears the selections for the current step.

✅ **Accept**

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point at Curve Middle Ribbon (for topological points in the Molded Forms task)

Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.**1** - Specifies the curve on which the point is placed.
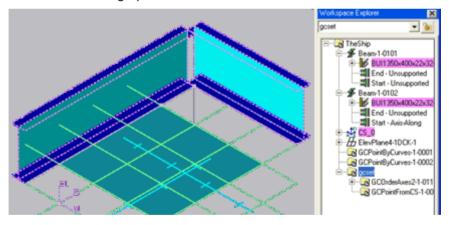
(2)

Specifies a surface on which to project the point. This value is optional.

Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

- ▫ 　1　 - No background when no input is selected.

- ▫ 　1　 - Yellow background when an input is selected.

- ▫ 　1　 - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

🗐 NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- `1` and `0.00 m ▼` - No background when the value of the original geometric construction is retained.

- `1` and `0.10 m ▼` - Yellow background when you change a value.

- `1` and `0.10 m ▼` - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- `1` - Blue background when the input is the same for all selected geometric constructions.

- `1` - No background with pink underline when the input is different for *all* selected geometric constructions.

- `1` - Blue background with pink underline when the input is different for *some* selected geometric constructions.

- `1` - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.

- `▼` - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.

  Y

  `▼`

  | 2.00 m [from GCOSRect400x300x100-1-0101, GCOSRect400x300x100-1-0102, GCOSOval500... |
  | 3.00 m [from GCOSCircle300-1-0001, GCOSCircle300-1-0002, GCOSCircle300-1-0003, G... |
  | Shared... |

❎ **Reject**

Clears the current selections.

✅ **Accept**

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.

- **More** - Select a system in the model.

📄 **NOTE** The **New GC Set** option is not used for topological points.

What do you want to do?

Place a point at the middle of a curve

1. Click **PointAtCurveMiddle** on the **Geometric Construction Palette**.

2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

   A preview of the point appears at the middle of the curve.

3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointAtCurveMiddle** in the **Geometric Construction Explorer**.*

4. If needed, create additional geometric constructions.

5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place a topological point at the middle of a curve

1. In the Molded Forms task, select **Insert** > **Topological Points**.

2. Click **PointAtCurveMiddle** on the **Point Palette**.

3. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

   A preview of the point appears at the middle of the curve.

4. Click **Finish**.

   *Point geometry is displayed in the graphic view and as **PointAtCurveMiddle** in the **Workspace Explorer**.*

Point at Curve Start

Tooltip: PointAtCurveStart

Creates a point at the beginning of a curve.

Required Input	Output
Curve, line, or edge	Point
Optional Input	
Projection surface	

Point at Curve Start Ribbon

 Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

 Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

✕ **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 3D again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ **IMPORTANT** Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the curve on which the point is placed.

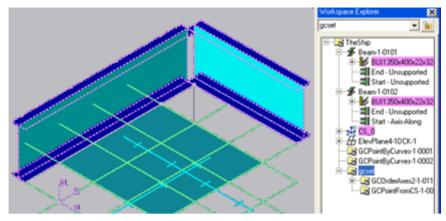(2) - Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- ⬛ 1 - No background color when there is no input.

- ⬛ 1 - Yellow background when an input is selected.

- ⬛ 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

▤ **NOTE** Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- [1] and [0.00 m ▼] - No background color when you keep the value of the original geometric construction.

- [1] and [0.10 m ▼] - Yellow background when you change a value.

- [1] and [0.10 m ▼] - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

Clears the selections for the current step.

✅ **Accept**

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place a point at the start of a curve* (on page)

- *Modify a geometric construction* (on page)

- *Change a geometric construction to a different definition* (on page)

- *Delete a geometric construction* (on page)

Place a point at the start of a curve

1. Click **PointAtCurveStart** on the **Geometric Construction Palette**.

2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

   A preview of the point appears at the beginning of the curve.

3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointAtCurveStart** in the **Geometric Construction Explorer**.*

4. If needed, create additional geometric constructions.

5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point by Curve and Surface

Tooltip: PointByCurveAndSurface

Creates a point at the intersection of a curve and a surface.

Required Input	Output
Curve, line, or edge	Point
Surface, reference plane, or plate system	
Optional Input	
Projection surface	

Point by Curve and Surface Ribbon

⬡ **Select**

   Selects an existing geometric construction object, such as a curve or surface, to edit or

delete.

✚ Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

✗ Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 🗇 again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:
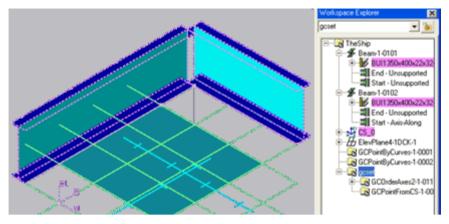
- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

    ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the curve.

2 - Specifies the surface, reference plane, or plate system. The surface must intersect the curve.

(3) - Specifies a tracking point used to control the direction of point placement when an ambiguity exists. A tracking point is only needed with some curves, such as:

- A circle defined by two points and a mid-point in which there are two arcs between the start and end points.

- A curve on multiple sides of a non-planar surface, such as a seam or reference curve wrapping around a hull.

This value is optional.

📝 NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

(4) - Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.

- 1 - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.

- 1 and 0.10 m - Yellow background when you change a value.

- 1 and 0.10 m - Blue background when the software changes a value to a new suggested value.

Reject

Clears the selections for the current step.

Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Track Flag - Controls how the tracking point ③ is used when an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?

- *Place a point at a curve and surface intersection* (on page)
- *Modify a geometric construction* (on page 267)
- *Change a geometric construction to a different definition* (on page 268)
- *Delete a geometric construction* (on page 268)

Place a point at a curve and surface intersection

1. Click **PointByCurveAndSurface** on the **Geometric Construction Palette**.

2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

3. Select a surface, reference plane, or plate system intersecting the curve.

   A preview of the point appears at the intersection.

4. If more than one solution is possible, click [3], and select the needed value for **Track Flag**.

   NOTE If changing the value for **Track Flag** does not give the needed solution, you may need to first create a point that you then select after clicking [3]. The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

5. Optionally, click [4] and select a plane.

   A preview of the point projected onto the plane appears.

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointByCurveAndSurface** in the **Geometric Construction Explorer**.*

7. If needed, create additional geometric constructions.

8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

   "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

   The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point by Curves

Tooltip: PointByCurves

Creates a point at the intersection of two curves.

Required Input	Output
Intersecting curves, lines, or edges	Point
Optional Input	
Projection surface	

Point by Curves Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first curve.

2 - Specifies the second curve. The second curve must intersect the first curve.

(3) - Specifies a tracking point used to control the direction of point placement when an ambiguity exists. A tracking point is only needed with some curves, such as:

- A circle defined by two points and a mid-point in which there are two arcs between the start and end points.

- A curve on multiple sides of a non-planar surface, such as a seam or reference curve wrapping around a hull.

This value is optional.

NOTE The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

(4) - Specifies a surface on which to project the point. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.

- 1 - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.

- 1 and 0.10 m - Yellow background when you change a value.

- ▪ ⬚1⬚ and ⬚0.10 m ▼⬚ - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

Clears the selections for the current step.

✅ **Accept**

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

⭐ **IMPORTANT** The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Track Flag - Controls how the tracking point ⬚3⬚ is used when an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?

- ▪ *Place a point at the intersection of two curves* (on page 245)
- ▪ *Modify a geometric construction* (on page 267)
- ▪ *Change a geometric construction to a different definition* (on page 268)
- ▪ *Delete a geometric construction* (on page 268)

Place a point at the intersection of two curves

1. Click **PointByCurves** on the **Geometric Construction Palette**.
2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.
3. Select a curve intersecting the first curve.

   A preview of the point appears at the intersection.

4. If more than one solution is possible, click ⬚3⬚, and select the needed value for **Track Flag**.

   📝 **NOTE** If changing the value for **Track Flag** does not give the needed solution, you may need to first create a point that you then select after clicking ⬚3⬚. The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

5. Optionally, click ⬚4⬚ and select a plane.

   A preview of the point projected onto the plane appears.

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointByCurves** in the **Geometric Construction Explorer***.

7. If needed, create additional geometric constructions.

8. Click **Close** to return to the main command ribbon.

★ **IMPORTANT**

▪ If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

   "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

   The message displays only during creation of an object and is unavailable when editing an object.

▪ The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point by Surfaces

Tooltip: PointBySurfaces

Creates a point at the intersection of a three surfaces.

Required Input	Output
Intersecting surfaces, reference planes, or plate systems	Point

Point by Surfaces Ribbon

⌕ Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

✚ Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

✕ Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

> Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 🔛 again on the main command ribbon.

Geometric Construction

> Displays the current geometric construction interface. You can also select:
>
> - A different, recently-used interface.
>
> - **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.
>
>   ★ **IMPORTANT** Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1 - Specifies the first surface, reference plane, or plate system.

2 - Specifies the second surface. The second curve must intersect the first surface.

3 - Specifies the third surface. The second curve must intersect the first and second surfaces.

📄 **NOTE** The intersection of the three surfaces must create a discrete point.

(4) - Specifies a tracking point used to control the direction of point placement when an ambiguity exists. A tracking point is only needed with some curves, such as:

- A circle defined by two points and a mid-point in which there are two arcs between the start and end points.

- A curve on multiple sides of a non-planar surface, such as a seam or reference curve wrapping around a hull.

This value is optional.

📄 **NOTE** The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.
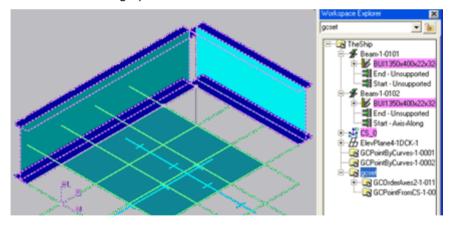
Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- `1` - No background color when there is no input.

- `1` - Yellow background when an input is selected.

- `1` - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

📄 **NOTE** Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- `[1]` and `[0.00 m ▼]` - No background color when you keep the value of the original geometric construction.

- `[1]` and `[0.10 m ▼]` - Yellow background when you change a value.

- `[1]` and `[0.10 m ▼]` - Blue background when the software changes a value to a new suggested value.

❌ Reject

Clears the selections for the current step.

✅ Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Track Flag - Controls how the tracking point **(3)** is used an ambiguity exists. Select **Near** to use the curve nearest to the tracking point. Select **Far** to use the curve farthest from the tracking point.

What do you want to do?
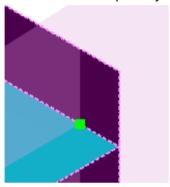
- *Place a point at the intersection of three surfaces* (on page 249)
- *Modify a geometric construction* (on page 267)
- *Change a geometric construction to a different definition* (on page 268)
- *Delete a geometric construction* (on page 268)

Place a point at the intersection of three surfaces

Click **PointBySurfaces** on the **Geometric Construction Palette**.

1. Select a surface, reference plane, or plate system.

2. Select a surface, reference plane, or plate system intersecting the first surface.

3. Select a surface, reference plane, or plate system intersecting the first and second surfaces, and forming a corner.

   A preview of the point appears at the intersection. In the example below, the point is at the intersection of two plate systems and a reference plane.

4. If more than one solution is possible, click **(4)**, and select the needed value for **Track Flag**.

   NOTE If changing the value for **Track Flag** does not give the needed solution, you may need to first create a point that you then select after clicking **(4)**. The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

5. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointBySurfaces** in the **Geometric Construction Explorer**.*

6. If needed, create additional geometric constructions.

7. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Point from Coordinate System

Tooltip: PointFromCS

Creates a point at the defined X, Y, and Z distances from the origin of the selected coordinate system or from a selected point.

Required Input	Output
Coordinate system	Point
Optional Input	
Reference point	
Parameters	
X, Y, and Z distances	

Point from Coordinate System Ribbon (for geometric constructions)

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 🔐 again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ IMPORTANT Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the coordinate system.

⟨2⟩

Specifies a reference point from which X, Y, and Z distances are measured. If no reference point is selected, the X, Y, and Z distances are measured from the origin of the coordinate system. This value is optional.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- 1 - No background color when there is no input.

- 1 - Yellow background when an input is selected.

- 1 - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- 1 and 0.00 m - No background color when you keep the value of the original geometric construction.

- 1 and 0.10 m - Yellow background when you change a value.

- ▪ **1** and **0.10 m** ▾ - Blue background when the software changes a value to a new suggested value.

❌ Reject

Clears the selections for the current step.

✅ Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

⭐ **IMPORTANT** The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

X

Specifies the distance along the X-axis from the origin or reference point.

Y

Specifies the distance along the Y-axis from the origin or reference point.

Z

Specifies the distance along the Z-axis from the origin or reference point.

Point from Coordinate System Ribbon (for topological points in the Molded Forms task)

📇 Properties

Opens the **Properties** dialog box for the selected geometric construction. This option is only available after selecting an existing geometric construction.

Geometric Construction

Displays the current geometric construction interface. You can also select a different, recently-used interface.

More

Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available. For more information, see *[Product Folder]*\Programming\Help\GeometricConstructions.chm, available when Programming Resources is installed.

1

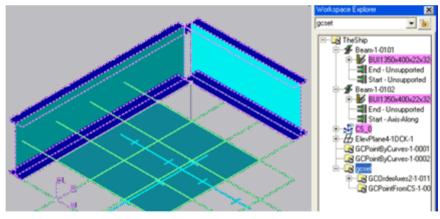Specifies the coordinate system.

(2)

Specifies a reference point from which X, Y, and Z distances are measured. If no reference point is selected, the X, Y, and Z distances are measured from the origin of the coordinate system. This value is optional.

Color Coding

Creating

For a new geometric construction, you can identify the status of input items by the background color of the step:

- ▪ $\boxed{1}$ - No background when no input is selected.

- ▪ $\boxed{1}$ - Yellow background when an input is selected.

- ▪ $\boxed{1}$ - Blue background when an input was selected from the previous geometric construction and can be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

Editing

When modifying an existing geometric construction to a different definition, the colors are used differently:

- ▪ $\boxed{1}$ and $\boxed{0.00\ m \quad \blacktriangledown}$ - No background when the value of the original geometric construction is retained.

- ▪ $\boxed{1}$ and $\boxed{0.10\ m \quad \blacktriangledown}$ - Yellow background when you change a value.

- ▪ $\boxed{1}$ and $\boxed{0.10\ m \quad \blacktriangledown}$ - Blue background when the software changes a value to a new suggested value.

Multi-editing

When modifying multiple existing geometric constructions and an input or parameter is available for all selections:

- ▪ $\boxed{1}$ - Blue background when the input is the same for all selected geometric constructions.

- ▪ $\boxed{1}$ - No background with pink underline when the input is different for *all* selected geometric constructions.

- ▪ $\boxed{1}$ - Blue background with pink underline when the input is different for *some* selected geometric constructions.

- - Yellow background with pink underline when all inputs that were originally the same have been changed. Remaining values have not been changed.

- - No background and no text displayed when the value of the parameter is different for the selected geometric constructions. Each value, along with the applicable geometric constructions, are available as selections in the box, as shown in the following example.

Reject

Clears the current selections.

Accept

Accepts the current selections, and displays a preview.

Finish

Completes geometric construction definition. The ribbon is still displayed, and parameters for additional geometry can be defined.

Name

Specifies the name of the geometric construction.

Parent

Specifies a parent for the geometric construction. Select one of the following:

- The current model. This is the default value.

- **More** - Select a system in the model.

NOTE The **New GC Set** option is not used for topological points.

X

Specifies the distance along the X-axis from the origin or reference point.

Y

Specifies the distance along the Y-axis from the origin or reference point.

Z

Specifies the distance along the Z-axis from the origin or reference point.

What do you want to do?

- *Place a point by distances from a coordinate system origin* (on page 255)
- *Place a point by distances from a reference point* (on page 256)
- *Modify a geometric construction* (on page 267)
- *Change a geometric construction to a different definition* (on page 268)
- *Delete a geometric construction* (on page 268)

- *Place a topological point by distances from a coordinate system origin* (on page 257) (in the Molded Forms task)

- *Place a topological point by distances from a reference point* (on page 257) (in the Molded Forms task)

- *Modify a topological point* (on page 227) (in the Molded Forms task)

- *Change a topological point to a different point definition* (on page 227) (in the Molded Forms task)

- *Delete a topological point* (on page 228) (in the Molded Forms task)

Place a point by distances from a coordinate system origin

1. Click **PointFromCS** on the **Geometric Construction Palette**.

2. Select a coordinate system.

3. Type values for **X**, **Y**. and **Z**.

   *A green preview point appears. It is offset from the coordinate system origin by the **X, Y.** and **Z** values.*

4. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and in the **Geometric Construction Explorer**.*

5. If needed, create additional geometric constructions.

6. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place a point by distances from a reference point

1. Create the point to use as a reference point. The point can be created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

2. Click **PointFromCS** on the **Geometric Construction Palette**.

3. Select a coordinate system.

4. Click ⑵, and select the reference point.

5. Type values for **X**, **Y**, and **Z**.

   A green preview point appears. It is offset from the reference point by the X, Y, and Z values and along the X, Y, and Z axes of the coordinate system.

6. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and in the **Geometric Construction Explorer**.*

7. If needed, create additional geometric constructions.

8. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place a topological point by distances from a coordinate system origin

1. In the Molded Forms task, select **Insert** > **Topological Points**.

2. Click **PointFromCS** on the **Point Palette**.

3. Select a coordinate system.

4. Type values for **X**, **Y**. and **Z**.

   *A green preview point appears. It is offset from the coordinate system origin by the **X, Y**. and **Z** values.*

5. Optionally, type a name for the point.

6. Optionally, select **More** in the **Parent** box, and select a parent system for the point.

7. Click **Finish**.

   Point geometry is displayed in the graphic view and in the Workspace Explorer.

Place a topological point by distances from a reference point

1. Create a point to use as a reference point by creating another topological point or using **Insert** > **Control Point**.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu) in the *Molded Forms User's Guide*

2. In the Molded Forms task, select **Insert** > **Topological Points**.

3. Click **PointFromCS** on the **Point Palette**.

4. Select a coordinate system.

5. Click (2), and select the reference point.

6. Type values for **X**, **Y**, and **Z**.

   *A green preview point appears. It is offset from the reference point by the **X, Y**, and **Z** values and along the **X, Y**, and **Z** axes of the coordinate system.*

7. Optionally, type a name for the point.

8. Optionally, select **More** in the **Parent** box, and select a parent system for the point.

9. Click **Finish**.

Point geometry is displayed in the graphic view and in the Workspace Explorer.
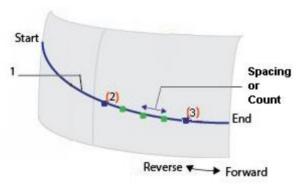
Points Along Curve

Tooltip: PointsAlongCurve

Creates an evenly-spaced pattern of points along a curve. Points are placed by defining the spacing between points or the total number of points.

Required Input	Output
1 - Curve, line, or edge	Points
Optional Input	
(2) - Start point	
(3) - End point	
Parameters	
Point spacing or point count	

Points Along Curve Ribbon

Select

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

Add

Selects geometry that defines a new geometric construction. This option is used with the

numbered step options listed below.

✕ Delete

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 🏲 again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

- A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ **IMPORTANT** Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies the curve on which the points are placed.

(2)

Specifies the starting point for the point pattern. This value is optional.

(3)

Specifies the ending point for the point pattern. This value is optional.

(4)

Specifies a tracking point used to control the direction of point placement an ambiguity exists. A tracking point is only needed with some curves, such as:

- A circle defined by two points and a mid-point in which there are two arcs between the start and end points.

- A curve on multiple sides of a non-planar surface, such as a seam or reference curve wrapping around a hull.

This value is optional.

📄 **NOTE** The selected tracking point must be an existing point previously created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- ▪ `1` - No background color when there is no input.

- ▪ `1` - Yellow background when an input is selected.

- ▪ `1` - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

NOTE Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- ▪ `1` and `0.00 m ▼` - No background color when you keep the value of the original geometric construction.

- ▪ `1` and `0.10 m ▼` - Yellow background when you change a value.

- ▪ `1` and `0.10 m ▼` - Blue background when the software changes a value to a new suggested value.

❎ Reject

Clears the selections for the current step.

✅ Accept

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

Direction

Specifies the direction of point creation. Select one of the following:

- **Normal** - Points are created from the start point to the end point.
- **Reversed** - Points are created from the end point to the start point.

Method

Specifies the way points are placed. Select one of the following:

- **By Spacing** - Points are placed at the distance specified in the **Spacing** box.
- **As Divisors** - Points are placed by the value in the **Count** box. The points are spaced equally along the distance between the start and end points.

Spacing

Specifies the distance between each point. This option is only available when **By Spacing** is selected in the **Method** box.

Count

Specifies the number of points to place. This option is only available when **As Divisors** is selected in the **Method** box.

Debug

When set to **On**, displays a sphere around each point to improve visibility. The size of the sphere is set by the value of **Radius**.

- Sphere preview:
- Sphere after point is created:

Radius

Specifies the sphere radius for **Debug**.

What do you want to do?

- *Place points along a curve by spacing with start and end points* (on page)
- *Place points along a curve by spacing without start and end points* (on page)
- *Place points along a curve by count with start and end points* (on page)
- *Place points along a curve by count without start and end points* (on page)
- *Modify a geometric construction* (on page)
- *Change a geometric construction to a different definition* (on page)
- *Delete a geometric construction* (on page)

Place points along a curve by spacing with start and end points

1. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

2. Create the points needed as start and end points on the curve using **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

3. Click **PointsAlongCurve** on the **Geometric Construction Palette**.

4. Select the same curve.

5. Click [2], and select a start point for the point pattern.

6. Click [3], and select an end point for the point pattern.
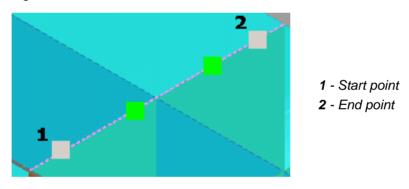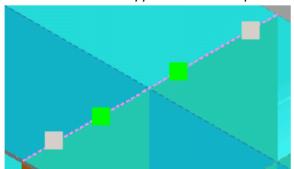
7. In the **Method** box, select **By Spacing**.

8. In the **Direction** box, select **Normal**.

9. In the **Spacing** box, type a value for the distance between points.

   A preview of the points appears. Spacing starts from the selected start point on the curve and continues until the remaining curve length to the selected end point is less than the spacing value.

1 - Start point
2 - End point

10. If the point spacing does not start from the needed end of the curve, select **Reversed** in the **Direction** box.

   Points start from the opposite end in the preview.

11. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*

12. If needed, create additional geometric constructions.

13. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

   "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

   The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place points along a curve by spacing without start and end points

1. Click **PointsAlongCurve** on the **Geometric Construction Palette**.

2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

3. In the **Method** box, select **By Spacing**.

4. In the **Direction** box, select **Normal**.

5. In the **Spacing** box, type a value for the distance between points.

   A preview of the points appears. Spacing starts from the system-defined start point of the selected curve and continues until the remaining curve length is less than the spacing value.
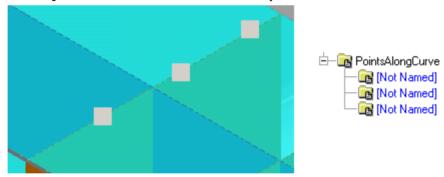
6. If the point spacing does not start from the needed end of the curve, select **Reversed** in the **Direction** box.

    Points start from the opposite end in the preview.

7. Click **Continue**.

    *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*

8. If needed, create additional geometric constructions.

9. Click **Close** to return to the main command ribbon.

★ IMPORTANT

-   If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

    "The previous execution of the command was not completed. Do you want to recover the 3D construction?"
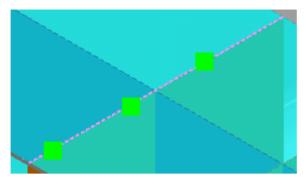
The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

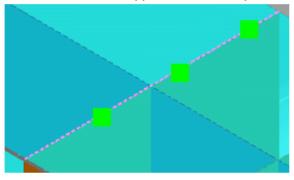## *Place points along a curve by count with start and end points*

1. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

2. Create the points needed as start and end points on the curve. These can be points created with **Insert** > **Control Point**, **Insert** > **Topological Points**, or other geometric construction point interfaces.

   Control Point in the *Common User's Guide*

   Topological Points (Insert Menu)

   Points (Geometric Construction Palette Dialog Box)

3. Click **PointsAlongCurve** on the **Geometric Construction Palette**.

4. Select the same curve.

5. Click $\boxed{2}$, and select a start point for the point pattern.

6. Click $\boxed{3}$, and select an end point for the point pattern.
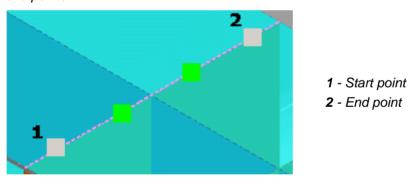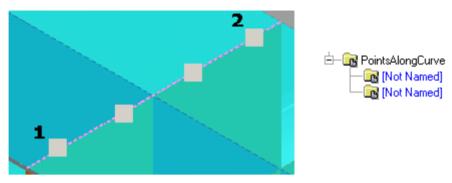
7. In the **Method** box, select **As Divisors**.

8. In the **Count** box, type the number of points to place.

   A preview of the points appears. The points are evenly spaced between the selected start and end points.

1 - Start point
2 - End point

9. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > **[Not Named]** in the **Geometric Construction Explorer**.*

10. If needed, create additional geometric constructions.

11. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

  "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

  The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Place points along a curve by count without start and end points

1. Click **PointsAlongCurve** on the **Geometric Construction Palette**.

2. Select a curve, such as a profile landing curve, a plate edge, or a geometric construction curve.

3. In the **Method** box, select **As Divisors**.

4. In the **Count** box, type the number of points to place.

   A preview of the points appears. The points are evenly spaced between the system-defined start and end points of the curve.

5. Click **Continue**.

*Temporary geometry is displayed in the graphic view and as **PointsAlongCurve** > [Not Named] in the **Geometric Construction Explorer**.*

6. If needed, create additional geometric constructions.

7. Click **Close** to return to the main command ribbon.

★ IMPORTANT

▪ If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

"The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

▪ The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.

Modify a geometric construction

Use this procedure to modify a geometric construction that is displayed after clicking **Continue** on a geometric construction ribbon but before clicking **Close** on the geometric construction ribbon.

1. On the geometric construction ribbon, click **Select** .

    ⚠ **WARNING** Do not click **Select** on the vertical toolbar. This removes you from your current command and deletes all temporary geometric constructions.

2. In a graphic view or the **Geometric Construction Explorer**, select the geometric construction to modify.

    *The geometric construction highlights in the graphic view, the ribbon for the geometric construction appears, and inputs highlight in the **Workspace Explorer**.*

3. Make the required changes.

4. Click **Accept** .

    A preview of the changed geometric construction appears.

5. Click **Continue**.

    Temporary geometry is displayed in the graphic view.

6. If required, create or modify additional geometric constructions.

Change a geometric construction to a different definition

Use this procedure when the geometric construction is displayed after clicking **Continue** on a geometric construction ribbon but before clicking **Close** on the geometric construction ribbon.

1. On the geometric construction ribbon, click **Select** .

   WARNING Do not click **Select** on the vertical toolbar. This removes you from your current command and deletes all temporary geometric constructions.

2. In a graphic view or the **Geometric Construction Explorer**, select the geometric construction to modify.

   The geometric construction highlights in the graphic view, the ribbon for the geometric construction appears, and the inputs highlight in the **Workspace Explorer***.*

3. In the **Macro Definition** box, click **More**.

   The **Select Geometric Construction** *dialog box appears showing the appropriate geometric constructions. For example, if the original geometric construction is a point, then only points appear.*

4. Select the needed geometric construction, and then click **OK**.

   The new selection appears in the **Macro Definition** *box with a yellow background. Options with blue backgrounds contain new values suggested by the software. Options with white backgrounds retain the values from the previous geometric construction.*

5. Make the required changes to options on the ribbon.

   Changed options appear with a yellow background.

6. Click **Accept** .

   A preview of the changed geometric construction displays.

7. Click **Continue**.

   Temporary geometry is displayed in the graphic view.

8. If required, create or modify additional geometric constructions.

Delete a geometric construction

Use this procedure to delete a geometric construction that is displayed after clicking **Continue** on a geometric construction ribbon but before clicking **Close** on the geometric construction ribbon and then **Finish** on the main command ribbon.

1. On the geometric construction ribbon, click **Select** .

   WARNING Do not click **Select** on the vertical toolbar. This removes you from your current command and deletes all temporary geometric constructions.

2. In a graphic view or the **Geometric Construction Explorer**, select the geometric construction to delete.

   The geometric construction highlights in the graphic view, the ribbon for the geometric construction appears, and inputs highlight in the **Workspace Explorer***.*

3. On the geometric construction ribbon, click **Delete** .

The geometric construction is deleted.

4. If required, click **Select** 🖱 and delete additional geometric constructions.

5. If required, click **Add** ➕ to create or modify additional geometric constructions.

Vertices from Curve or Surface

Tooltip: VerticesFromCurveOrSurf

Creates points at the vertices of a curve or surface, including profile and seam landing curves and plate systems. A point is created at each end point, corner point, each change in tangency of 1° or more on a curve, and at each location where a curve crosses a surface patch boundary.

Required Input	Output
Plate part or profile part	Points

Vertices from Curve or Surface Ribbon

🖱 **Select**

Selects an existing geometric construction object, such as a curve or surface, to edit or delete.

➕ **Add**

Selects geometry that defines a new geometric construction. This option is used with the numbered step options listed below.

✕ **Delete**

Deletes a selected geometric construction.

Cancel

Cancels the macro, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon.

Close

Verifies the validity of values selected for the options, closes the geometric construction ribbon and the **Geometric Construction Palette** dialog box, and then returns to the main command ribbon. The selected values are still available when you click 🔳 again on the main command ribbon.

Geometric Construction

Displays the current geometric construction interface. You can also select:

▪ A different, recently-used interface.

- **More** - Opens the **Select Geometric Construction** dialog box. For a new geometric construction, all interfaces delivered with the software are available. For an existing geometric construction, only similar interface types are available.

  ★ **IMPORTANT** Geometric constructions displaying only in the **Select Geometric Construction** dialog box are not intended for general usage. You should first consult your Intergraph support representative before using these geometric constructions.

1

Specifies a surface or curve.

Color Coding

For a new geometric construction, you can identify the status of input items by the background color of the step:

- `1` - No background color when there is no input.

- `1` - Yellow background when an input is selected.

- `1` - Blue background when an input was selected for the previous geometric construction and the input can also be used for the current geometric construction.

📄 **NOTE** Selected inputs appear highlighted in pink in the **Workspace Explorer** and as pink dotted lines in the graphic view:

When changing an existing geometric construction to a different definition, colors represent the different value changes:

- `1` and `0.00 m ▼` - No background color when you keep the value of the original geometric construction.

- `1` and `0.10 m ▼` - Yellow background when you change a value.

- `1` and `0.10 m ▼` - Blue background when the software changes a value to a new suggested value.

❌ **Reject**

Clears the selections for the current step.

✅ **Accept**

Accepts the current selections and displays a preview.

Continue

Completes the geometric construction definition. The ribbon continues to display so that you can define parameters for additional geometry.

★ IMPORTANT The geometric constructions are not created and saved to the model until you click **Finish** on the main command ribbon to create the model object.

What do you want to do?

- *Place points on vertices* (on page)

- *Modify a geometric construction* (on page 267)

- *Change a geometric construction to a different definition* (on page 268)

- *Delete a geometric construction* (on page 268)

Place points on vertices

1. Click **VerticesFromCurveOrSurf** on the **Geometric Construction Palette**.

2. Select a plate part or profile part.

   Green preview points at all vertices are visible.

3. Click **Continue**.

   *Temporary geometry is displayed in the graphic view and as **VerticesFromCurveOrSurf** > **[Not Named]** in the **Geometric Construction Explorer**. In the example below, a point is place at each corner and at the surface patch boundary of a bounding plate.*

4. If needed, create additional geometric constructions.

5. Click **Close** to return to the main command ribbon.

★ IMPORTANT

- If you stop the command before clicking **Finish**, the software does not save the geometric constructions to the model. When you restart the command, the following message displays, giving you the option to recover the geometric constructions:

   "The previous execution of the command was not completed. Do you want to recover the 3D construction?"

The message displays only during creation of an object and is unavailable when editing an object.

- The geometric constructions are not saved to the model until you click **Finish** on the main command ribbon to create the model object.
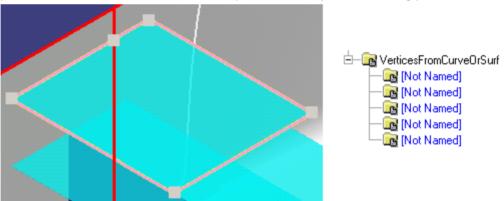
Geometric Construction Explorer

Displays geometry related to the selected geometric construction interface. The geometry is added after you define parameters for the interface and click **Continue** on the **Geometric Construction** ribbon.

When creating a new object, such as a curve or a surface, the geometry is temporary and is not named.

Geometry used to create the object is saved with the object and named. When modifying an existing object, the geometry names display.

The **Geometric Construction Explorer** displays parameters in a hierarchical list similar to the **Workspace Explorer**. Selecting a geometry name highlights the geometry in the graphic view.

Commands

The following commands are available by right-clicking an item in the hierarchical list:

Show

Turns on the graphical display of a hidden object in the graphic view.

Hide

Turns off the graphical display of an object in the graphic view. The names of hidden objects display in italics in the **Geometric Construction Explorer**.

Export

Exports the selected geometric construction to an XML file. This command is only available when you right-click the top level geometric construction set.

Import

Imports a previously exported geometric construction from an XML file into the workspace. The imported geometric construction replaces any existing geometric construction. This command is only available when you right-click the top level geometric construction set.

📝 **NOTE** **Export** and **Import** are primarily intended for use by administrators creating complex geometric construction macros. The commands can also be used to save multiple geometric

constructions, such as a set of seams or profile stiffener landing curves on a hull, that you may not complete in a single session, or that you want to reuse in a similar model.
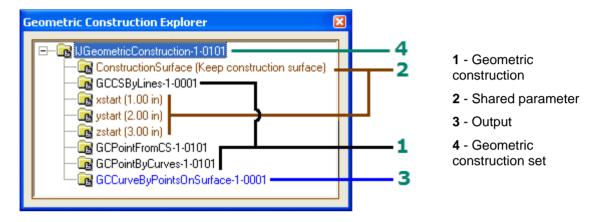
Color Coding

You can identify the purpose of objects by the text color:

Black text - An individual geometric construction macro.
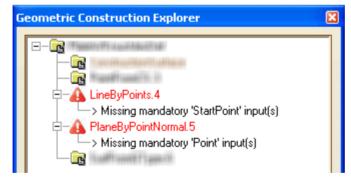
Red text - A shared parameter used by multiple geometric constructions.

Blue text - Output geometry from the macros.

1 - Geometric construction

2 - Shared parameter

3 - Output

4 - Geometric construction set

Error Display
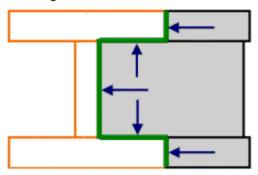
When a required input for a geometric construction is deleted, the **Geometric Construction Explorer** displays an error, such as those shown in the following example.

S E C T I O N   1 9

Place Member Generic Assembly Connection

Places an assembly connection between a member part and a bounding part. This command creates the assembly connection, applies the appropriate flange and web cuts, and creates the appropriate physical connections. A typical example is the bounding of a member to the flange and web faces of another member.

NOTE Member generic assembly connections are typically created automatically when a member is placed with **Place Linear Member Systems** and connected to a detailed plate system or edge reinforcement. You can use **Place Member Generic Assembly Connection** in situations where the connection is not created automatically.

The member is bounded to ports at the faces or edges of the bounding part. The ports used can be edited after creating the assembly connection. This command supports the following situations:

- A member bounded by a plate edge port.

- A member bounded by a plate face port.

- A member bounded by a flange edge port.

- A member bounded by a web or flange face port.

NOTES

- The selected bounding geometry must be on detailed parts.

- This command can generate physical connections to a single plate part.

Place Member Generic Assembly Connection Ribbon

Displays the options that you use to place an assembly connection between a built-up member part and intersecting face ports.

Properties

Activates the **Assembly Connection Properties** dialog box, which you use to view and modify the properties of the assembly connection. For more information, see *Assembly*

Connection Properties Dialog Box (on page 289).

Supported Member

Select the built-up member for which to create the assembly connection.

Supporting (Bounding) Geometry

Select a bounding port that intersects the end of the supported member to bound. If necessary, select additional bounding ports to which to create assembly and physical connections. Only those bounding ports that intersect the selected bounded end of the supported member are available for you to select.

Finish

Places the assembly connection.

Cancel

Clears the current selection.

Accept

Accepts the current selection. After you accept the supporting member and bounding geometry, the **Properties** button becomes active, and the **Assembly Connection** list is populated with the available connection types.

Assembly Connection

Select an assembly connection type from the list.

Rule Based

Enables or disables the **Assembly Connection** list. Selecting this check box disables **Assembly Connection** list.

Modify Member Generic Assembly Connection Ribbon

Displays the options that you use to modify an existing assembly connection between a built-up member part and intersecting face ports.

Properties

Activates the **Assembly Connection Properties** dialog box, which you use to view and modify the properties of the assembly connection. For more information, see *Assembly Connection Properties Dialog Box* (on page 289).

Supported Member

Displays the built-up member for which to create the assembly connection. The supported member cannot be modified.

Supporting (Bounding) Geometry

Select a bounding port that intersects the end of the supported member to bound. If necessary, select additional bounding ports to which to create assembly and physical connections. Only those bounding ports that intersect the selected bounded end of the supported member are available for you to select.

Finish

Places the assembly connection.

Cancel

Clears the current selection.

Accept

Accepts the current selection. After you accept the supporting member and bounding geometry, the **Properties** button becomes active, and the **Assembly Connection** list is populated with the available connection types.

Assembly Connection

Select an assembly connection type from the list.

Rule Based

Enables or disables the **Assembly Connection** list. Selecting this check box disables the **Assembly Connection** list.

What do you want to do?

▪ *Place a member generic assembly connection* (on page)

▪ *Modify a smart occurrence object* (on page 58)

▪ *Delete a smart occurrence object* (on page 58)

▪ *Edit assembly connection ports* (on page 278)

Place a member generic assembly connection

1. Click **Place Member Generic Assembly Connection**.

2. Select the member for which to create the assembly connection.

3. Select the bounding ports.

    TIPS

    ▪ The software creates physical connections between the supported member and the bounding ports that you select.

    ▪ Only those ports that intersect the selected end of the supported member are available for selection.

    ▪ If you select only the top flange of the bounding built up, the software may not be able to completely trim the bounded member.

4. Click **Accept** .
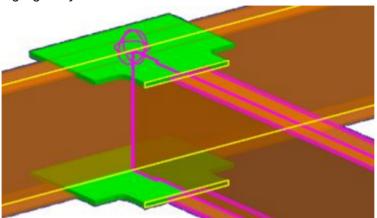
    NOTE The **Rule Based** check box must be clear to select the assembly connection type.

5. Select the assembly connection type from the **Assembly Connection** list.

6. Click **Finish**.

Edit assembly connection ports

1. In a graphic view or in the **Workspace Explorer**, select a member with an assembly connection that you want to modify.

   The member and assembly connection highlight in magenta. The assembly connection ports highlight in yellow.

2. Select the port that you want to change.

   *The **Modify Member Generic Assembly Connection** ribbon appears with **Supporting (Bounding) Geometry** ⬛ as the active option.*

3. To remove an existing port, select it.

   The port is no longer yellow.

4. To add a new port, select it.

   The port highlights in yellow.

5. Click **Accept** ✅.

6. If you want to change the assembly connection type select a new value from the **Assembly Connection** list.

7. Click **Finish**.
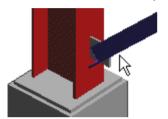
📝 NOTES

- A list of members and plates connected to the selected member can be viewed on the **Relationship** tab of the **Assembly Connection Properties** dialog box. Select a connected object from the list to highlight it in the graphic view.

- Because you can modify the bounding geometry ports, but not the bounded member, the **Supported Member** ⬛ option is not available.

Place Assembly Connection

 Places an assembly connection or free end cut at the selected frame connection. Assembly connections define the necessary trimming between member parts and provide for the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections also control cutbacks, copes, notches, bolt holes, weld preparations, and slots.

This command places member assembly connections or a free end cut on the member. The software automatically places slab assembly connections when you place the slab.
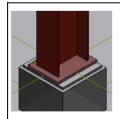
Member Assembly Connections

Member assembly connections are similar to frame connections, but define the necessary trimming between member parts and the generation of parts such as base plates, gusset plates, and clip angles. Assembly connections control member features including cutbacks, copes, notches, bolt holes, and slots. Whether or not features are placed depends on the member assembly connection type and the geometry of the connection between the members.

There are several basic assembly connections delivered with the software. You can create your own assembly connections by editing the **StructAssemblyConnections.xls** workbook, and then bulk loading the workbook. For more information on creating your own assembly connections, see the Structure Reference Data Guide.

NOTE For very complex nodal connections, instead of writing a custom assembly connection, you might want to try *Trim Members* in the Structure task.

The following general assembly connections are delivered with the software:
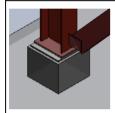
	The base plate assembly connection places a plate at the end of an unsupported member. This assembly connection requires an unsupported frame connection on one member. For example, use this assembly connection to place a base plate at the bottom of a column.
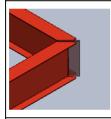
	The corner gusset plate assembly connection connects a vertical brace to a beam and column intersection using a gusset plate. This assembly connection requires a frame connection with three members, such as vertical corner brace.
	The fitted assembly connection connects two members. This assembly connection requires a frame connection with two members, such as axis, seated, or flush. Examples of this connection include a beam framing into a column or a beam framing into another beam.
	The gusset plate assembly connection connects a vertical or horizontal brace to a beam or a vertical brace to a column using a gusset plate. This assembly connection requires a frame connection with two members, such as axis.
	The miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection requires a frame connection with two members, such as axis. In addition, the members must be end connected.
	The splice assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The general surface assembly connection connects a member end with a nonmember surface such as a slab, a deck, or a hull plate. The member is cut to surface and a base plate is placed on the member end.
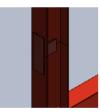
End Cut Assembly Connections

The following end cut assembly connections are delivered with the software:

	The end cut along assembly connection cuts the end of one member to the sides or bottom of another member along the second member's length.
	The end cut bearing plate assembly connection connects the end of a member along the axis of another member. This is typically used to attach the end of a member to the flange of the supporting member. The bearing plate connection works at any angle and provides circular, rectangular, and triangular bearing plates depending on the supported member's shape.
	The end cut miter assembly connection connects two members that meet at an angle but are co-planar. This assembly connection required a frame connection with two members, such as axis. In addition, the members must be end connected.
	The end cut split assembly connection connects two members that are collinear and end connected. This assembly connection requires a frame connection with two members, such as axis.
	The end cut split long box assembly connection is used when the supported and supporting members are perpendicular to each other. The supporting member is cut square to the supported member while an item from the Axis category is placed on the supported member.
	The end cut split seam angle assembly connection is used when the supported and supporting members are: ▪ member axes are collinear to each other ▪ have identical cross-section type and size ▪ have identical cardinal point and rotation values
	The end cut split short box assembly connection is used when the supported and supporting members are perpendicular to each other. The supported member is cut square to the supporting member while an item from the Axis category is placed on the supporting member.

End Cut Assembly Naming Conventions

Each end cut assembly connection name consists of three parts:
[EndCutCase]_[WebCutType]_[FlangeCutType]

where:

- [EndCutCase]

  - **Axis** - Supported member is bounded by the axis curve of the supporting member

  - **LongBox** - End-to-end perpendicular supported member is extended, and an axis end cut is applied to the supporting member

  - **ShortBox** - End-to-end perpendicular supporting member is extended, and an axis end cut is applied to the supported member

  - **Miter** - End-to-end non-collinear

  - **Split** - End-to-end collinear

  - **SeamAngle** - End-to-end collinear where one end is extended

- [WebCutType]

  The WebCutType is used to type of web cut that is to be applied at the top of the web and at the bottom of the web. The following combinations are available (W indicates a straight cut, C indicates a Cope cut, and S indicates a Snipe cut):
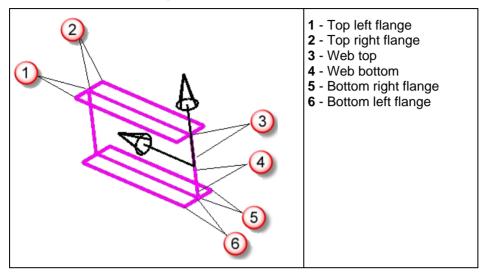
  - W#W#

  - W#C#

  - W#S#

  - C#C#

  - C#S#

  - C#W#

  - S#S#

  - S#C#

  - S#W#

- [FlangeCutType]

  The FlangeCutType is used to type of flange cut that is to be applied at the left flange section and at the right flange section. The following combinations are available (W indicates a straight cut, C indicates a cope cut, and S indicates a snipe cut):

  - W#W#

  - W#S#

  - S#S#

  - S#W#

You cannot apply different flange cuts when the supported member has both a top flange and a bottom flange. In these cases, the same flange cut type is applied to both the top and bottom flange.
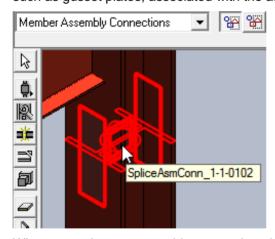
You cannot apply different web cuts or flange cuts in the end-to-end cases. The end-to-end cases place a web cut and flange cut on both the supported member and the supporting member. For these cases, the same web cut and flange cut are applied to both the supported member and the supporting member.

	1 - Top left flange **2** - Top right flange **3** - Web top **4** - Web bottom **5** - Bottom right flange **6** - Bottom left flange

See *Members* in the Structure task for important related information.

Locating Assembly Connections

Assembly connections do not display in the model. However, if you set the **Locate Filter** to **Member Assembly Connections** in the Structure task, you can locate and select assembly connections for review and editing. Assembly connections are located at the ends of member parts and appear as circles when you move the cursor over them. Any assembly components, such as gusset plates, associated with the assembly connection also highlight.

When you select an assembly connection, the software displays the assembly connection type in the ribbon. Select the **Edit > Properties** command to edit the assembly connection properties. You cannot measure distances using the assembly connection graphic as an end point when using the **Tools > Measure** command.

Place Assembly Connection Ribbon

Specifies the properties for the assembly connection that you are placing or editing.

Assembly Connection Properties

Activates the **Assembly Connection Properties** dialog box. You can use this dialog box to specify additional properties that you cannot set on the ribbon. For more information, see *Assembly Connection Properties Dialog Box* (on page 289).

Select Member/Connection

Activated automatically by the software so that you can select the frame connection or the member for which you want to place assembly connections.

Finish

Click to place the assembly connection the model.

Cancel

Rejects the selected object.

Accept

Confirms that the selected members, is the member to place assembly connections for. The software displays in tentative mode the results of the assembly connection.

Condition

Specifies how you want the software to handle existing assembly connections when you try to place a new assembly connection at the same location. Select **Retain existing** to keep the existing assembly connection. Select **Update existing** to replace the existing assembly connection with the new assembly connection.

By Rule

Select to allow the software to select the assembly connection to use based on the selected member parts and their orientation to each other.

Type

Select the assembly connection type to use. If you select **By Rule**, the software determines the correct assembly connection to use based on the geometry between the member parts, and this option is unavailable. If you select **More**, all available assembly connections display from which you can select the assembly connection to use. For more information about assembly connections, see Member Assembly Connections.

Member Part Parent

Select to make the member part the parent object of the assembly connection. Clear this option to specify a generic system, using the **System** box, to be the assembly connection parent. Assembly connections created using member parts as their parents can be edited to have a system parent later if needed.

System

Select the system to which the assembly connection belongs. You can define new systems in the Systems and Specifications task. This option is unavailable if you select **Member Part Parent**. Assembly connections created using a system as their parent can be edited to have a member part as their parent later if needed.

Name

Specify the name of the assembly connection.

Modify Auxiliary Parts Ribbon

Displays the options that you use to add auxiliary parts to the selected assembly connection.

Properties

Activates the **Assembly Connection Properties Dialog Box**, which you use to view and modify the properties of the assembly connection before the assembly connection is committed to the database. For more information, see *Assembly Connection Properties Dialog Box* (on page 289).

Assembly Connection Smart Item

Displays valid smart items for the selected assembly connection.

Add/Delete Auxiliary Parts

Select the parts that can be connected to the selected assembly connection and do not already have an existing assembly connection. Parts that do not have common geometry with the selected assembly connection cannot be selected.

Accept

Adds parts from selected parts to the assembly connection and saves the changes to the database.

Reject

Removes any selected parts from the auxiliary parts list. Only selections made after the last commit are removed.

What do you want to do?

- *Place an assembly connection* (on page 286)
- *Edit assembly connection properties* (on page 287)
- *Delete an assembly connection* (on page 287)
- *Add auxiliary parts to an assembly connection* (on page 287)
- *Edit assembly connection parts* (on page 288)
- *Place free end cuts* (on page 288)
- *Edit free end cut properties* (on page 288)
- *Delete free end cut* (on page 288)
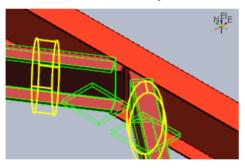
Place an assembly connection

1. Click **Place Assembly Connection** ⬚ on the vertical toolbar.

2. Select the **By Rule** option if you want the software to automatically select the type of assembly connection to place. Clear the **By Rule** option to select the assembly connection type yourself.

3. Select the frame connection nearest the member end to which to apply the assembly connection.
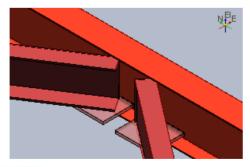
4. If you are selecting the assembly connection type yourself, use the **Type** option.

5. Click **Accept** ✓.

   The software automatically selects the assembly connection and displays the results.
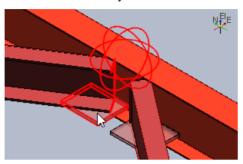
6. Click **Finish**.

📄 NOTE For more information about the types of assembly connections, see Member Assembly Connections.

Edit assembly connection properties

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter**.

3. Select the assembly connection to edit.

4. Click **Edit > Properties**.

5. Edit the assembly connection properties as needed. For more information, see *Assembly Connection Properties Dialog Box* (on page 289).

Delete an assembly connection

1. Click **Select** ⬚ on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter**.

3. Select the assembly connection to delete.

4. Click **Delete** ✗.

Add auxiliary parts to an assembly connection

This command supports adding parts to an existing assembly connection by selecting an adjacent plate, profile or member parts that have overlapping geometry.

1. Select an existing assembly connection.

2. Click **Select Auxiliary Parts** 🗔 on the ribbon bar.

   📝 NOTE The **Select Auxiliary Parts** 🗔 is enabled for any assembly connection. You can only add valid auxiliary parts to an assembly connection.

3. Click **Accept** ✅ to add all intersecting parts on the selected parts to the assembly connection.

Edit assembly connection parts

1. Select an assembly connection, and click **Select Auxiliary Parts** on the ribbon bar.

2. Select the parts not already connected to the selected assembly connection that have intersecting geometry.

3. To remove a single part, re-select it from the graphic view or **Workspace Explorer**.

4. To remove the auxiliary parts added in the current command execution, click **Reject** on the ribbon bar.

5. Click **Accept** to add all intersecting parts on the selected parts to the assembly connection.

Place free end cuts

1. Click **Place Assembly Connection** on the vertical toolbar.

2. Select the unsupported frame connection nearest the member end to which you want to apply the free end cuts.

   If you select a member part, the software automatically places free end cuts at the unsupported frame connections.

3. Click **Accept.**

   The software automatically selects the free end cuts and displays the results.

4. Click **Finish**.

Edit free end cut properties

1. Click **Select** on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter** box.

3. Select the free end cut to edit.

4. Click **Edit > Properties**.

5. Edit the free end cut properties as needed. For more information, see *Free End Cut Properties Dialog Box* (on page 418).

Delete free end cut

1. Click **Select** on the vertical toolbar.

2. Select **Member Assembly Connections** in the **Locate Filter**.

3. Select the free end cut to delete.

4. Click **Delete** .

Assembly Connection Properties Dialog Box

Specifies the properties for the member assembly connection that you are editing.

Occurrence Tab (Assembly Connection Properties Dialog Box) (on page 289)
Definition Tab (Assembly Connection Properties Dialog Box) (on page 293)
Relationship Tab (on page 400)
Configuration Tab (on page 401)
Notes Tab
Selection Tab (Assembly Connection Properties Dialog Box) (on page 294)

See Also
Edit assembly connection properties (on page 287)

Occurrence Tab (Assembly Connection Properties Dialog Box)

The **Occurrence** tab displays the member assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Name

Displays the name of the assembly connection. The assembly connection name is based on the **Name Rule** selection. To type a new name for the assembly connection, in the **Name Rule** box, select **User Defined**, and then type a name for the assembly connection in the **Name** box.

Name Rule

Specify the naming rule to use to name this assembly connection.

- **Default Name Rule** - Names the assembly connection using this format: <assembly connection>-<location>-<unique index> where <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, GussetPlateAsmConn_1-1-0045.

- **Unique Name Rule** - Names the assembly connection using this format: <parent system name>-<assembly connection>-<location>-<unique index> where <parent system name> is the parent System selected for the assembly connection, <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, Structure System-GussetPlateAsmConn_1-1-0045.

- **User Defined** - Select to specify the assembly connection name yourself in the **Name** box.

System

Select the system to which the assembly connection that you are placing belongs. You can

create new systems in the Systems and Specifications task.

Base Plate Assembly Connection Properties

Depth Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension C in the figure.

Width Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension D in the figure.

Sizing Rule

Select the sizing rule method for the base plate.

Plate Category

Select the plate category.

Plate Type

Select the plate type.

Miter Assembly Connection Properties

Top Distance

Specifies the distance between the top flange of the member section and the top of the plate.

Bottom Distance

Specifies the distance between the bottom flange of the member section and the bottom of the plate.

Left Distance

Specifies the distance between the left edge of the member section and the left edge of the plate.

Right Distance

Specifies the distance between the right edge of the member section and the right edge of the plate.

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select

Center to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

With Plates

Specifies whether or not a plate is inserted between the member ends. Select **False** to not place the plate. Select **True** to place the plate.

Clearance

Specifies the distance between the member ends.

Sizing Rule

Select the sizing rule method for the base plate.

Slab by Member Boundary

Clearance

Type a clearance distance between the edge of the slab and the member.

Port Face Position

Select the location on the member at which the slab is to stop. You can select the outmost plane, the centerline, or the in-most plane on the member.

Detailed Connection

Select **True** to trim the slab using the boundary member as the cutting edge. Select **False** to not trim the slab.

Offset

Specify the distance between the selected **Port Face Position** and the edge of the slab. A negative value moves the edge into the body of the slab. A positive value moves the edge out from the body of the slab.

Slab by Slab Assembly

Trim Type

Defines the type of trimming.

- **By Whole Slab** - Trim by the slab boundary geometry before trim.

- **By Lateral Face** - Trim by the slab boundary geometry after trim.

- **By Extended Lateral Face** - Trim by the extension of the lateral face of the slab boundary.

Trim Excess Material

Specify whether to trim the exceeding part.

Trim Excess Material Offset

Specify the offset applied on the lateral face used to trim.

Slab Free Edge Assembly

Reference Direction

Select the reference direction for the angle:

Normal

The angle is measured from a vector perpendicular to the slab edge.

- **Horizontal** - The angle is measured from the global XY plane in the model.
- **Vertical** - The angle is measured from the Z-Axis in the model.

Angle

Type a slope for the slab edge represented by the assembly connection. If the slab was place using the **Face Position Top**, the side face rotates about the top slab edge. If the slab was placed using the **Face Position Bottom**, the side face rotates about the bottom slab edge.

Offset

Specify the distance between the selected boundary object and the edge of the slab.

Splice Assembly Connection Properties

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

Clearance

Specifies the distance between the ends of the members.

Splice With

Select the plates that you want to use in the splice.

Web Plate Position

Select a web plate position.

Distance from flange gage line

Specifies the distance from the flange gage line.

Distance from web gage line

Specifies the distance from the web gage line.

Flange Plate Thickness

Specifies the thickness of the flange plates.

Flange Plate Length

Specifies the length of the flange plates.

Flange Plate Width

Specifies the width of the flange plates.

Flange Plate Category

Select the plate category for the flange plates.

Flange Plate Type

Select the plate type for the flange plates.

Web Plate Thickness

Specifies the thickness of the web plates

Web Plate Length

Specifies the length of the web plates.

Web Plate Width

Specifies the width of the web plates.

Web Plate Category

Select the plate category for the web plates.

Web Plate Type

Select the plate type for the web plates.

General Surface Assembly Connection Properties

With Pad

Select **True** to place a steel pad between the member end and the surface. Select **False** to have the member end connect directly to the surface.

Pad Type

Select the shape of the pad.

Sizing Rule

Select whether you want to the software to automatically size the pad or if you want to size the pad.

Offset

Type the distance between the end of the member and the face of the surface.

See Also
Assembly Connection Properties Dialog Box (on page 289)

Definition Tab (Assembly Connection Properties Dialog Box)

The **Definition** tab displays the member assembly connection properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection and then selected the properties command, only the common properties between the selected assembly connections display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also
Assembly Connection Properties Dialog Box (on page 289)

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the member assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

♥ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  📝 **NOTE** If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

★ IMPORTANT

- The available questions, answers, and results vary depending upon the detailed parts associated with the connection.

- The questions in each **Selection** tab represent the default rules delivered with the software. Customized rules may have different questions.

Place Manual Physical Connections

Creates a physical connection between two parts. You can manually create connections between part types such as detailed plate parts, detailed profile parts, member parts, slabs, walls, or any combination of these parts.

The two methods to create manual physical connections are:

1. Part and parts (default option)
2. Part and ports

Place Manual Physical Connections Ribbon - Part and Parts Method

Properties

Activates the **Physical Connection Properties** dialog box, which you use to view and modify the properties of the physical connection that you are about to place in the model. For more information, see *Physical Connection Properties Dialog Box* (on page 424).

Command Mode

Activates the part and parts method.

Select Base Part

Selects the first part.

Select Parts to Create Physical Connections with

Selects the adjoining parts to create connections.

Finish

Places the physical connections.

Make 1st Part Parent

Places the physical connection with the first object selected (parent base part). If this is not selected, then the physical connection is placed with parts selected in the second step (**Select Parts to Create Physical Connections with**).

Weld Class

Specifies the physical connection weld class (or weld item) that the physical connection is based on.

Weld Item

Specifies the physical connection weld item if the weld class allows multiple items.

NOTE You must always select the weld class. After you select the correct class and click **Compute/Preview**, the **Weld Item** list contains eligible weld items.

Rule Based

Enables or disables the **Weld Item** list. Selecting this check box disables the **Weld Item** list.

Place Manual Physical Connections Ribbon - Part and Ports Method

Properties

Activates the **Physical Connection Properties** dialog box, which you use to view and modify the properties of the physical connection that you are about to place in the model. For more information, see *Physical Connection Properties Dialog Box* (on page 424).

Command Mode

Activates the part and ports method.

Select Parent Part

Select the first part to be used in the physical connection. This first part is also used by default as the parent part of the physical connection.

Select Port to be connected from Parent Part

Select an edge or surface port from the parent part. This port is used as the first input to all physical connections to be created.

Select Other Port(s) to be connected to

Select an edge or surface port from another part that overlaps the port from the parent part. You can select multiple ports to create multiple physical connections based on the selected port from the parent part.

Finish

Places the physical connections.

Parent

Specifies the parent system for the physical connections.

Weld Class

Specifies the physical connection weld class (or weld item) that the physical connection is based on.

Weld Item

Specifies the physical connection weld item if the weld class allows multiple items.

What do you want to do?

- *Place a manual physical connection - part and parts method* (on page 297)
- *Place a manual physical connection - part and ports method* (on page 297)
- *Modify a smart occurrence object* (on page 58)
- *Delete a smart occurrence object* (on page 58)

Place a manual physical connection - part and parts method

1. Click **Place Manual Physical Connection**.

2. Click **Command Mode,** and select the part and parts method .

3. Select the target part.

4. Select the surrounding parts.

5. Select a value for **Weld Class**.

6. Click **Compute/Preview**.

    NOTE The **Rule Based** check box must be clear to select a value for the weld item.

7. If needed, select a value for **Weld Item**.

8. Click **Finish**.

    One or more physical connections are created between the parts.

NOTES
- You can only select parts that have overlapping geometry and do not currently have a physical connection.

- You can select multiple parts if identical physical connections (by weld type) are created on the selected ports.

Place a manual physical connection - part and ports method

1. Click **Place Manual Physical Connection** .

2. Click **Command Mode,** and select the part and ports method .

3. Select the part that contains the first port for physical connection.

4. Select the port on the first part.

5. Select the overlapping port from another part.

6. Select the parent system for **Parent**.

7. Select a value for **Weld Class**.

    NOTE The **Rule Based** check box must be clear to select a value for the weld item.

8. If needed, select a value for **Weld Item**.

9. Click **Finish**.

    One or more physical connections are created between the parts containing the selected

ports.

NOTES

- You can only select parts that have overlapping geometry and do not currently have a physical connection.

- You can select multiple parts if identical physical connections (by weld type) are created on the selected ports.

S E C T I O N   2 2

Import Ship Structure

File > **Import** > **Ship Structure**, available in the Molded Forms and Structural Detailing tasks, imports Tribon marine structure XML data into Smart 3D for planning and outfitting work.

To use this command, you must purchase and install the **Intergraph Smart**™ **3D Tribon Interface** (SEBY812AG) from Smart 3D. Please contact Intergraph Support for licensing information.

Before you can import the data into Smart 3D, you must export the data from Tribon Initial Design Version M3 (Version 1.3). Smart 3D only supports XML in Schema V3 format.

Import Ship Structure Dialog Box (on page)

Import ship structure data

1. Export data from Tribon Initial Design to create an M3 XML file.
2. In Smart 3D, select **File** > **Import** > **Ship Structure**.

   The **Import Ship Structure** *dialog box appears.*
3. Click the button next to the **XML file or path** box to navigate to the appropriate folder location, and do one of the following:

   Click **Open** to process all of the valid XML files in the folder.

   OR

   Select the individual XML file to use for import, and then click **Open**.
4. Click the button next to the **Map file** box to select the XLS map workbook to use for import, and then click **Open**.
5. Select a system to use as the parent for the imported data.
6. Click **OK** to import the Tribon data as specified, or click **Submit Job** to run the import as a batch job.

   📄 NOTE For more information about setting up the batch service, see *Installing Batch Services* in the *Intergraph Smart*™ *3D Installation Guide.*

Import Ship Structure Dialog Box

Sets options for importing Tribon XML data.

XML file or path

Specify the name of and path to the XML file exported from Tribon. To process multiple files simultaneously, specify the name of and path to the folder that contains the XML files to import. Alternatively, you can use the browse feature and navigate to the file or folder of files to be used for import.

Map file

Specify the name of and path to the mapping file between the Tribon XML data and the Smart 3D data in the catalog. For more information, see *Import Translation Map Workbook* (on page 300).

System

Select a system in the Smart 3D model under which to place the imported data.

Import Translation Map Workbook

The Microsoft Excel workbook named **SM_Mapping.xls** is delivered in the *[Reference Data Folder]*\SharedContent\Data\TribonTranslator\MappingFiles folder. This workbook maps Tribon material and cross-section names to Smart 3D material and cross-section names.

The Cross Section worksheet maps profile cross-section names between the Tribon XML file and Smart 3D.

Cross Section Worksheet (on page 300)

The Material worksheet maps Tribon material names to Smart 3D material names.

Material Worksheet (on page 301)

Cross Section Worksheet

The **Cross Section** sheet of the **SM_Mapping.xls** workbook maps Tribon profile cross-section names in the XML file to Smart 3D profile cross-section names.

Section Name

Type the Smart 3D profile cross-section name.

Imported Family Name

Type the Tribon profile cross-section name as it appears in the XML file.

NOTE The last item listed for each section type is the default value for that section type.

Material Worksheet

The **Material** sheet of the **SM_Mapping.xls** workbook maps Tribon materials in the XML file to Smart 3D materials.

Material Type

Type the Smart 3D material type.

Material Grade

Type the Smart 3D material grade.

Imported Material Grade

Type the Tribon material name as it appears in the XML file.

Repairing Tribon Data

The Tribon translator software includes a command that you can use to repair the data in Tribon XML files. Examples of data to repair include:

- Instance properties

- Profile part and seam location

★ **IMPORTANT** The integrity of the source data is the responsibility of the generating software (in this case, Tribon). When you repair the data with this command, you assume liability and responsibility for the results. Intergraph does not assume any liability for the data modifications.

See Also
Repair Tribon XML (Tools Menu) (on page 302)

Repair Tribon XML (Tools Menu)

Tools > **Repair Tribon XML** modifies data in Tribon XML files. You can then import the modified data into Smart 3D.

Repair Tribon XML Dialog Box **(on page 303)**

Repair Tribon data

1. Import Tribon data from an XML file. For more information, see *Import Ship Structure* (on page 299).

2. If you see errors in the imported geometry, perform the following steps.

3. Click **Tools** > **Repair Tribon XML**.

4. On the **Repair Tribon XML** dialog box, specify the input file, output file, and log file for the repair process.

5. Select the curved panels to repair.

6. Check the repair options as needed.

7. Click **OK**.

8. Re-import the data using the repaired XML file. Objects that are different from the previous import are automatically updated in the model.

Repair Tribon XML Dialog Box

Sets options for repairing Tribon data.

Files

Input XML file

Specify the import file.

Output XML file

Specify the output file. You can import this file into Smart 3D using the **File** > **Import** > **Ship Structure** command.

Log File

Specify a log file that stores information about the repair process.

Panel

Curved Panels

Select this box to choose curved panels, which are panels defined on a sculpted, not flat, surface.

Plane Panels

This option is not available in this release.

Select the curved panels to be modified in the xml

Click the panels to repair.

Options

Change instance property of panel to As Defined

Changes the instance property to **AsDefined**. This property specifies the position of objects relative to their defining geometry. The different values for this property can be:

- **AsDefined** - The objects are located where they were defined.

- **Reflected** - If the objects were defined on the port side, they are actually on the starboard side. If the objects were defined on the starboard side, they are actually on the port side.

- **Both** - The objects are symmetrical with respect to the origin and are located on both sides.

Move profile parts to other side of center line (Change Y value)

Reflects the profile parts about the Y-axis so the parts are in the correct location.

Move seams to other side of centerline (Change Y value)

Reflects the seams about the Y-axis so the parts are in the correct location.

The other options in this section are not available in this release of the software.

SECTION 24

Manufacturing Service Manager

Manufacturing Service Manager (in the Structural Manufacturing task) or **Tools** > **Manufacturing Service Manager** reviews and updates manufacturing objects. **Review** displays a report of the current condition of the manufacturing objects that you have selected. **Updating** pulls any outstanding changes into the manufacturing environment. The command is available on the vertical toolbar in the Structural Manufacturing task, and by clicking **Tools** > **Manufacturing Service Manager** in the Molded Forms, Structural Detailing, Planning, and Structural Manufacturing tasks.

You can select individual parts, assemblies, or blocks for this command.

The **Manufacturing Service Manager** pulls changes into the manufacturing environment instead of associatively pushing from other tasks into the manufacturing environment. In this way, the manufacturing environment is similar to the drawing environment; you have control over which objects get updated and when.

NOTES

- A manufacturing object goes on the **To Do List** for the following reasons:
    - The object has a nonworking status, or it contains objects which are included in different permission groups. You must change the manufacturing part status back to **Working**, and then update the manufacturing part by using the **Manufacturing Service Manager**.
    - The parent system of the manufacturing object was split in the Molded Forms task, and that split migration produced an error.
    - A compute found a failure within the rule and the manufacturing plate, manufacturing profile, or manufacturing member returned a warning.
    - Some other compute failure, with a specific action per entity, occurred.
- The same behavior occurs when the molded form split is performed by someone who does not have write permissions to the manufacturing part. In this case, a second person with write permission to the manufacturing part must update the manufacturing part by the **Manufacturing Service Manager**.

Manufacturing Service Manager Ribbon

Displays the controls used to update and review manufacturing objects.

Selection of Possible Report Data

Opens a dialog box from which you can set up reports.

Select

Selects objects in the model.

Finish

Starts the **Manufacturing Service Manager**.

Cancel

Clears the current selection.

Accept

Accepts the current selection.

Action

Determines whether the **Manufacturing Service Manager** updates the objects, reviews the objects, or reports the Detailed Review of the objects.

Filter

Specify the type of manufacturing part to use.

- **Parts** - All plate, profile, and member parts.
- **Parts or Assemblies** - All plate, profile, member, and different assembly types. For assemblies, you can select blocks, assembly blocks and assemblies.
- **Plate Parts** - Plate parts.
- **Plate Parts and Connected Profiles** - Includes all profiles connected to the plate parts.
- **Profile Parts** - Profile parts.
- **Member Parts** - Member parts.
- **Systems** - All plate systems, profile systems, and member systems.
- **Systems or Assemblies** - All systems and assemblies.
- **Plate Systems** - All plate systems.
- **Plate Systems and Connected Profiles** - All member systems.
- **Profile Systems** - All profile systems.
- **Member Systems** - All member systems and design member systems.

Connected

Specify which type of connected parts to use.

- **<Ignore>** - Only the selected part.
- **Direct Connected** - Includes objects that are physically connected to the selected part.
- **Indirect Connected** - Includes direct objects and objects that are physically connected to the direct objects.

What do you want to do?

- *Review a manufacturing object* (on page)
- *Update a manufacturing object* (on page)
- *Review or update a set of objects selected by a filter* (on page)
- Detail Review a manufacturing object

Review a manufacturing object

1. Click **Select** 🔍 on the vertical toolbar.

2. Select the manufacturing objects to review.

   💡 **TIP** You can select the objects in the model or from the **Workspace Explorer**.

3. Click **Review** in the **Action** box.

4. Select the type of manufacturing object to review in the **Report Per** box.

5. Click **Finish**.

Update a manufacturing object

1. Click **Select** 🔍 on the vertical toolbar.

2. Select the manufacturing objects to review.

   💡 **TIP** You can select the objects in the model, or from the **Workspace Explorer**.

3. Click **Update** in the **Action** box.

4. Select the type of manufacturing object to review in the **Report Per** box.

5. Click **Finish**.

💡 **TIP** The software updates the database as each manufacturing object is updated. Clicking **Cancel** stops the update for any objects that have not yet been posted to the database.

Review or update a set of objects selected by a filter

1. Click **More** from the filter selection list.

   *The **Select Filter** dialog box displays.*

2. Set up the filter to meet your requirements. For example, you could define a filter to select plate and profile parts in a specific block.

   The software finds and highlights the objects that match the filter.

3. Click **Service Manager** 📧 (in the Structural Manufacturing task) or **Tools** > **Manufacturing Service Manager**.

   The software prompts you to use the selected list of objects.

4. Click **Yes**.

5. If required, add additional objects to the list.

6. Click **Review** or **Update** in the **Action** box.

7. Select the type of manufacturing object to review in the **Report Per** box.

8. Click **Finish**.

Review Dialog Box

Displays the controls used to review the current manufacturing status of manufacturing objects.

NOTE This dialog box opens when you click **Finish** on the **Manufacturing Service Manager Ribbon**.

The manufacturing object status display includes:

- **Up to Date** - Indicates that the object does not require updating.
- **Out of Date** - Indicates that the object requires updating.
- **Can Be Deleted** - Indicates that the object is no longer needed.
- **To-Do-List (Error)** - Indicates that the object is on the **To Do List** because of an error condition.
- **To-Do-List (Out-of-Date)** - Indicates that the object is on the **To Do List** because it requires updating.
- **To-Do-List (Warning)** - Indicates that the object is on the **To Do List** because of a warning condition.
- **Edited** - Indicates that the object has been edited with the part editor. If you modify this part by using the **Manufacturing Plate** or **Manufacturing Profile** commands, or use the **BO Recompute** command, then the status does not display as **Edited**.

Properties

Displays the **Selection of Possible Report Data** dialog box. For more information, see *Selection of Possible Report Data Dialog Box* (on page 308).

Report Per

Displays the current settings for the report data that is set to display. The information that displays depends on what you have set to show or hide on the **Selection of Possible Report Data** dialog box.

Update

Updates the objects. Only the objects shown on the dialog box are updated. This option does not update parts with a status of **Edited**.

Cancel

Closes the dialog box.

File name

Type the path and file name for the report file. You can use the **Browse** button beside the box to browse through the file system.

View

Displays the report file in the default text editor, such as **Notepad**.

Selection of Possible Report Data Dialog Box

Displays the controls used to determine which report data displays on the **Review** dialog box.

Attributes Tab

Option

Determines the manufacturing object type for which you are setting the review options. Select the box next to the option, and change the setting to **Show** or **Hide** as necessary.

Settings Tab

Review Settings

Turn on the settings to apply when the **Action** is set to **Review**. The available options include:

- Only include Out of Date manufacturing objects
- Only include manufacturing objects in **To Do List**
- Check detailed part in **To Do List**
- Include light parts
- Refresh after Update
- Highlight Out of Date objects

Update Settings

Turn on the settings to apply when the **Action** is set to **Update**. The available options include:

- Delete objects under light part
- Update objects with detailed part in **To Do List**
- Update manufacturing objects in **To Do List**
- Recompute all manufacturing objects

Copy by Family

The **Edit > Copy by Family** command replicates structural objects on the **Select Copy Family** dialog box using three methods:

- **Copy Similar** replicates major portions of the model with the ability to automatically update the replicas as changes are made. A copy similar family consists of groups of structure with similar grid planes. One group is copied to the other groups. A typical example is to define a transverse bulkhead, including profile stiffeners, seams, and openings, and copy the bulkhead to other transverse locations.

- **Copy Symmetry** replicates portions of the model across a longitudinal centerline symmetry grid plane. A copy symmetry family consists of groups of structure on both the port and starboard sides of the symmetry plane. One group is reflected about the symmetry plane to the other group.

- **Copy Across Models** replicates major portions of a model to another model.

📝 **NOTE** Within the **Create Target Group** and **Modify Target Group**, you can manually select objects to copy. For more information, see *Manually copy objects* (on page 324).

Copy by Family Batch Services

Smart 3D uses the SmartPlant Batch Services application to help you perform **Copy by Family** functions in the background. You can submit the batch jobs that are executed in a batch queue. For this, you must create, enable, and start batch queues before submitting the batch jobs.

📝 **NOTE** For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide.*

You can use the following guidelines listed to run the batch services:

- SmartPlant Batch Services must be installed on the client workstation that submits a batch job and on the batch server workstation that processes the job. The necessary software queues must also be created. For information on installing SmartPlant Batch Services and setting up the batch queues, see *Installing SmartPlant Batch Services* in the *Smart 3D Installation Guide.*

- The batch jobs that are submitted using this functionality can only be monitored by the SmartPlant Batch Services application.

- Use the **Intergraph Batch Manager** tool provided by the Intergraph Batch Services to discontinue any batch job. The **Batch Timeout** property specified on the **Property** page is no longer valid.

- The mapped account that performs the batch job does not need to be an administrator on the batch server.

- The batch processes run as a Windows service using a configured login. You do not need to be logged on to the batch server.

- In the **Intergraph Batch Server Properties (Local Computer)** > **Log On**, make sure **Allow service to interact with desktop** is selected.

- For Windows XP batch servers, grant Write permissions for each mapped account to the %SYSTEMDRIVE%/Documents and Settings/Default User/Application Data/Microsoft/AddIns folders.

- For 64-bit batch servers, the %windir%/SysWOW64/config/systemprofile/desktop folder must be manually created and every mapped account must have Write access on that folder. For most machines, the %windir% folder is be located in the C:/Windows. However this may vary depending on your Windows installation.

- For batch jobs, Microsoft Excel must have the same security settings that are necessary for interactive updates. Configure Microsoft Excel for every SmartPlant Batch Services mapped account on the batch server, so that it can be accessed by the batch processes. For more information, see *Reports User's Guide.*

- Microsoft Excel must be opened on the batch server at least once by each SmartPlant Batch Services mapped user. Also, **Trust Settings** for Microsoft Excel must be included.

- For Windows 7 batch servers, the registry key HKEY_USERS/.DEFAULT/Software/Microsoft/VBA must be manually created with Write permissions given individually for each mapped account.

Select Copy Family Dialog Box

Toolbar

Create New Similar Family

Creates a new copy similar family. For more information on creating copy similar families, see *Similar Family Properties Dialog Box* (on page 330).

Create New Symmetry Family

Creates a new symmetry family. For more information on creating copy symmetry families, see *Symmetry Family Properties Dialog Box* (on page 333).

Create New Across Model Family

Creates a new across model family. For more information on creating across model families, see *Across Model Family Definition Dialog Box* (on page 335).

Delete Family Definition

Deletes the selected family.

Rename Family Definition

Renames the selected family. You must select a family before you can rename that family.

Edit Family Definition

Edits the selected family. You must select a family before you can edit that family.

Refresh Family

Removes any relationships created in the family by using the **Synchronize** command and the **Copy by Family** ribbon. For example, **Refresh Family** restores highlighting of objects in the graphics view for the **Find and correlate objects** step if that step has been previously carried out.

Copy Shortcuts

Configures copy shortcut keys for similar or symmetry families and specify a log file to store the copy operation. For more information, see *Copy Shortcuts Settings Dialog Box* (on page 338).

Copy Family Hierarchy

Copy Across Model Families

Lists the Copy Across Model Families that you have defined.

Copy Similar Families

Lists the copy similar families that you have defined.

Copy Symmetry Families

Lists the copy symmetry families that you have defined.

Batch

Displays the **Copy by Family Batch** dialog box. You can use this dialog box to execute jobs in a batch queue. For more information, see *Copy By Family Batch Dialog Box* (on page 339).

Synchronize

Opens the **Copy by Family** ribbon for the selected family.

Close

Closes the dialog box.

Copy by Family Ribbon

The **Copy by Family** ribbon copies structural objects between the source and target family groups, and keeps the family groups in synchronization. Similar families and symmetry families both use the ribbon. The ribbon is available after clicking **Synchronize** on the **Select Copy Family** dialog box.

Find and correlate objects

Select to highlight similar or symmetrical structural objects that meet the criteria of both the **Source Group** and **Target Group** of the Similar or Symmetry family. When you click **Finish**, the software adds the objects to the copy family so that duplicates are not created during the copy. Select **Find Man** to manually add objects that were not found automatically.

The search for similar/symmetry objects first considers if they have the same inputs (that is, the same boundaries, supporting frame, and so on). It then matches geometry (for example, starboard plate is the same as the mirrored port plate). For similar relationships, parametric changes caused by the hull are recognized.

Because this is a strong matching criterion, multiple finds in the same group are typically caused by the same object being placed twice. These overlapping finds are reported as errors for you to fix, and the similar/symmetry relation is not made.

Because this is such a strong criteria, the **Manual Find** option allows you to force objects to be similar that are not similar according to the automatic criteria. After using **Manual Find**, you can run **Find and correlate objects** to match objects that use the manually similar

entities as inputs.

Use the **Find and correlate objects** option before a copy or modify operation if there is a chance that a similar/symmetry object has been created in the target area after initially using the **Copy by Family** command.

Potential similar or symmetry objects can be created in the target area by the following operations:

- Creating a new object in the target area.

- The copy result of a different family copies an object into the same target area.

- An automatic process creates a new object in the target area. For example, the execute split command creates intersection seams in the target area that are similar to the source area.

Create target group copy

Select to highlight objects in the source group that are not in the target group. The software automatically finds all plate systems that use the grid plane of the source group as the surface definition. The software then finds all children of the plate system (seams, stiffeners, openings, and features) and all the structural objects that are bounded by either the grid plane or the plate systems defined by the grid plane and all their children. You can click objects in the graphics view to clear them and prevent them from being copied from the source group to the target group. Select **Browser** to clear objects in a hierarchical list. The software displays the cleared objects in green in both the graphics view and the Browser. When you click **Finish**, all highlighted objects are copied to create the target group.

Modify target group copy

Select to highlight objects that are different between the source and target groups. If a family for which a copy has already been created contains changed copied objects or new objects, use this option to update the target group copy. You can click objects in the graphics view to clear them and prevent them from being updated. Select **Browser** to clear objects in a hierarchical list. The software displays the cleared objects in green in both the graphics view and the Browser. When you click **Finish**, all highlighted objects are updated in the target group.

Delete target group copy

Select to highlight objects that have been previously copied and deleted from the source group, but remain in the target group. You can click objects in the graphics view to clear them and prevent them from being deleted in the target group. The software marks these cleared objects as recognized differences between the groups and displays the objects in green. Select **Browser** to clear objects in a hierarchical list. The software displays the cleared objects in green in both the graphics view and the Browser. When you click **Finish**, all highlighted objects are deleted from the target group.

Finish

Performs the copy create, modify, or delete as appropriate.

Reject

Resets the selected objects.

Accept

Confirms that the selected objects are the ones you want to work on.

Family

Select the family to work with. You can create new families by using the **Create New Similar Family** 🛡️ or **Create New Symmetry Family** 🛡️ commands on the **Select Copy Family** dialog box. For more information, see *Select Copy Family Dialog Box* (on page 337).

Source Filter

Select the filter to be used by the create or modify operations.

💡 **TIPS**

- When you copy or update objects in a similar or symmetry family, the **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the Copy by Family operation is performed on only those objects satisfying both the predefined filter and the Workspace filter. When the **<Filter> + Workspace Filter** option is in use, the **Source Filter** box is highlighted in yellow, and the **Find** and **Delete** buttons are unavailable. The **<Filter> + Workspace Filter** option supports only similar and symmetry families.

- When you copy across models, you can select the **Select Filter** option to display all filters in the model corresponding to those in the source model of the selected copy across model family. Select the filter you need from the **Select Filter** dialog box. Your filter selection is remembered in the session file and is used as the default source filter the next time you copy across models. For more information, see *Select Filter Dialog Box* (on page 336).

Source Group

Select the family group to use as the copy source.

Target Group

Select one or more family groups to create or modify.

Browser

Opens a browser similar to the **Workspace Explorer** that lists objects included in the target group copy. You can clear objects from the browser. The **CDF** tab (Copy Dependent Full) lists the objects in a hierarchy. The **CDT** tab (Copy Dependent by Type) sorts the objects by type. You can de-select objects from the browser. The software displays selected objects in yellow and cleared objects in green.

🗐 **Family Synchronize Options**

Defines find, create, and modify options for synchronizing the family. For more information, see *Synchronize Options Dialog Box* (on page 329).

What do you want to do?

- *Create a copy similar family using a single grid plane* (on page 314)
- *Create a copy similar family using multiple grid planes* (on page 315)
- *Modify a copy similar family* (on page 316)
- *Delete a copy similar family* (on page 316)
- *Create a copy symmetry family* (on page 316)

- *Modify a copy symmetry family* (on page 317)
- *Delete a copy symmetry family* (on page 317)
- *Create a copy across models family using a single grid plane* (on page 317)
- *Create a copy across models family using multiple grid planes* (on page 318)
- *Create a copy across models family by range* (on page 318)
- *Modify a copy across model family* (on page 319)
- *Delete a copy across model family* (on page 319)
- *Copy objects in a similar family* (on page 319)
- *Update objects in a similar family* (on page 320)
- *Delete objects in a similar family* (on page 321)
- *Copy objects in a symmetry family* (on page 322)
- *Update objects in a symmetry family* (on page 323)
- *Delete objects in a symmetry family* (on page 324)
- *Manually copy objects* (on page 324)
- *Copy objects using Copy Shortcuts* (on page 325)
- *Copy objects in an across model family* (on page 326)
- *Find and correlate objects across models* (on page 326)
- *Select the source filter for copy across models* (on page 326)
- *Schedule a Copy by Family operation as a batch job* (on page 328)
- *Configure the Smart 3D batch queues* (on page 328)
- *Configure account mapping for batch services* (on page 328)

Create a copy similar family using a single grid plane

1. Click **Edit > Copy by Family**.
2. Click **Create New Similar Family** 🛡 in the **Select Copy Family** dialog box.
   *The **Similar Family Properties** dialog box displays.*
3. Type a unique name for the family in the **Name** box.
4. Select the definition method for the family: **Standard**, **By Range**, or **Overlapping**.
5. Select the box in the grid under the **.G 1** group heading.
6. In the **Workspace Explorer**, select the grid plane that defines the objects in **.G 1**.
7. Select the box in the grid under the **.G 2** group heading.
8. In the **Workspace Explorer**, select the grid plane that defines the location of **.G 2**.
9. Optionally, click **Insert group** if you need more than two groups in the family.
10. Define the grid planes for any new groups as you did for **.G 2**.

11. Click **OK**.

☰ NOTES

- By default, **.G 1** is the copy source group for the family (as indicated by the check in the group heading). You can specify another group to be the copy source by selecting the checkbox in that group's heading.

- All structural objects that are related to the plane in a group are included in the group after **Synchronize** is performed. For more information, see *Select Copy Family Dialog Box* (on page 337).

- You can select a group column heading box (for example, **.G 1**) and rename the group.

Create a copy similar family using multiple grid planes

1. Click **Edit > Copy by Family**.

2. Click **Create New Similar Family** 🔰 in the **Select Copy Family** dialog box.

   *The **Similar Family Properties** dialog box displays.*

3. Type a unique name for the family in the **Name** box.

4. Select the definition method for the family: **Standard**, **By Range**, or **Overlapping**.

5. Select the box in the grid under the **.G1** group heading.

6. In the **Workspace Explorer**, select the first grid plane that defines the objects in **.G1**.

7. Select the box in the grid under the **.G2** group heading.

8. In the **Workspace Explorer**, select the grid plane that defines the location of **.G2**.

9. Click **Insert corresponding item (row)**.

10. Select the new (second) box in the grid under the **.G1** group heading.

11. In the **Workspace Explorer**, select the second grid plane that defines the objects in **.G1**.

12. Select the new (second) box in the grid under the **.G2** group heading.

13. In the **Workspace Explorer**, select the second grid plane that defines the location of **.G2**.

14. Optionally, click **Insert group** if you need more than two groups in the family.

15. Define the grid planes for any new groups as you did for **.G2**.

16. Click **OK**.

☰ NOTES

- By default, **.G1** is the copy source group for the family (as indicated by the check in the group heading). You can specify another group to be the copy source by selecting the checkbox in that group's heading.

- All structural objects that are related to both planes in a group are included in the group after **Synchronize** is performed. For more information, see *Select Copy Family Dialog Box* (on page 337).

- You can select a group column heading box (for example, **.G1**) and rename the group.

Modify a copy similar family

1. Click **Edit > Copy by Family**.

    *The **Select Copy Family** dialog box displays.*

2. Select the family to modify under the **Copy Similar Families** 🛡 hierarchy.

3. Click **Edit Family Definition** 📑 in the **Select Copy Family** dialog box. This activates the **Similar Family Properties** dialog box.

4. Using the dialog box controls, edit the family as needed.

    ★ IMPORTANT

    *   The grid planes in a row cannot be modified. You can delete the group with grid planes to change, and add a new group.

    *   An existing row cannot be deleted. You can delete the family in the **Select Copy Family** dialog box and create a new one with the needed number of rows.

Delete a copy similar family

1. Click **Edit > Copy by Family**.

    *The **Select Copy Family** dialog box displays.*

2. Select the family to delete under the 🛡 **Copy Similar Families** hierarchy.

3. Click **Delete Family Definition** ✕.

Create a copy symmetry family

1. Click **Edit > Copy by Family**.

2. Click **Create New Symmetry Family** 🛡 in the **Select Copy Family** dialog box.

    *The **Symmetry Family Properties** dialog box.*

3. Type a unique name for the family in the **Name** box.

4. In the **Workspace Explorer**, select one or more coordinate systems to define the symmetry plane.

    *The coordinate systems display in the **Symmetry Planes** box.*

    All selected coordinate systems must have coincident Y=0 planes.

5. In the **Port Group Name** box, type a name for the port side group.

6. In the **Starboard Group Name** box, type a name for the starboard side group.

7. Under **Source Group**, select whether the port side or the starboard side is the copy source group.

8. Click **OK**.

Modify a copy symmetry family

1. Click **Edit > Copy by Family**.

   The Select Copy Family dialog box displays.

2. Select the family to modify under the **Copy Symmetry Families** hierarchy.

3. Click **Edit Family Definition** in the **Select Copy Family** dialog box.

   The Symmetry Family Properties dialog box displays.

4. Using the dialog box controls, edit the family as needed.

   ★ IMPORTANT An existing coordinate system in the **Symmetry Planes** box cannot be deleted. You can delete the family in the **Select Copy Family** dialog box and create a new one with the needed coordinate system.

Delete a copy symmetry family

1. Click **Edit > Copy by Family**.

   The Select Copy Family dialog box displays.

2. Select the family to delete under the **Copy Symmetry Families** hierarchy.

3. Click **Delete Family Definition** ✕.

Create a copy across models family using a single grid plane

1. Click **Edit** > **Copy by Family**.

2. Click **Create New Across Model Family** in the **Select Copy Family dialog** box.

   The Across Model Family Definition dialog box displays.

3. Type a name for the family in the **Name** box, or use the default name.

4. Type a value in the **Range Growth Value** box, or use the default value. The **Method** definition is selected as **Standard** by default.

5. To change the **Group Name**, click the cell corresponding to the target group name, and type the name.

6. To select the source ship, click the cell **Ship Name** of the source, and select the ship from the list. To change the **Group Name**, click the cell corresponding to the source group name, and type the name.

7. Click the cell **Corresponding Elements1** under the **Target** or **Source,** and select the grid plane that defines the target or source group.

8. Click **OK**.

   📄 NOTE All structural objects that are related to the plane in a group are included in the group after **Synchronize** is performed. For more information, see *Select Copy Family Dialog Box* (on page 337).

Create a copy across models family using multiple grid planes

1. Click **Edit** > **Copy by Family**.

2. Click **Create New Across Model Family** ⬚ in the **Select Copy Family dialog** box.

   *The **Across Model Family Definition** dialog box displays.*

3. Type a name for the family in the **Name** box, or use the default name.

4. Type a value in the **Range Growth Value** box, or use the available value. The **Method** definition is selected as **Standard** by default.

5. To change the **Group Name**, click the cell corresponding to the target group name, and type the name.

6. To select the source ship, click the cell **Ship Name** of the source, and select the ship from the list. To change the **Group Name**, click the cell corresponding to the source Group Name, and type the name.

7. Click the cell **Corresponding Elements1** under the **Target** or **Source,** and define the grid plane that defines the target/source group.

8. Click the **Add Corresponding Elements** button to add **Corresponding Elements** rows to define additional grid planes for the target or source group.

9. Click **OK**.

Create a copy across models family by range

1. Click **Edit** > **Copy by Family**.

2. Click **Create New Across Model Family** ⬚ in the **Select Copy Family** dialog box.

   *The **Across Model Family Definition** dialog box displays.*

3. Type a name for the family in the **Name** box, or use the default name.

4. Type a value in **Range Growth Value** box, or use the default value.

5. Select **Method** from the **By Range** list.

6. To change the **Group Name**, click the cell corresponding to the target group name, and type the name.

7. To select the source ship, click the cell **Ship Name** of the source, and select the ship from the list. To change the **Group Name**, click the cell corresponding to the source group name, and type the name.

8. Click the cells **Corresponding Elements1** and **Corresponding Elements2** under the **Target** or **Source,** and define the grid planes for the target or source group.

9. Click **Add Corresponding Elements** to add **Corresponding Elements** rows to define additional grid planes for the target/source group. Because there are only two corresponding elements allowed when the **Method** definition is **By Range**, the **Add corresponding Elements** and **Delete Corresponding Elements** buttons are not available.

10. Click **OK**.

Modify a copy across model family

1. Click **Edit** > **Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to modify under the **Copy Across Model Families** hierarchy.

3. Click **Edit Family Definition** in the **Select Copy Family** dialog box.

   *The **Copy Across Model Family Properties** dialog box displays.*

4. Using the dialog box controls, edit the family definition as needed.

   TIPS

   - You can edit the name of the family and the range growth value.

   - Optionally, you can add additional **Corresponding Elements** rows to define additional grid planes using **Add Corresponding Elements**.

5. Click **Apply** or **OK** to save the changes.

   IMPORTANT

   - You cannot change the **Method** definition.

   - The target and source group details cannot be edited.

   - For more information on configuration details, see *Across Model Family Definition Dialog Box* (on page 335).

Delete a copy across model family

1. Click **Edit** > **Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to delete under the **Copy Across Model Families** hierarchy.

3. Click **Delete Family Definition** .

Copy objects in a similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to work with under the **Copy Similar Families** hierarchy.

   TIP For instructions on creating a new copy similar family, see *Create a copy similar family using a single grid plane* (on page 314) and *Create a copy similar family using multiple grid planes* (on page 315).

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

> TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group when you defined the family. For more information, see *Similar Family Properties Dialog Box* (on page 330).

6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**. Optionally, select **Multiple** to copy to more than one target group.

7. Click **Find and correlate objects** .

   The software highlights all existing structural objects that are similar and meet the criteria of both the Source Group and the Target Group.

8. Click **Create target group copy** on the ribbon.

   The software highlights in yellow all the objects in the Source Group that are not in the Target Group.

9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to cyan.

10. Click **Finish**.

    The software copies the highlighted objects in the Source group to the Target group.

Update objects in a similar family

1. Click **Edit > Copy by Family**.

   The Select Copy Family dialog box displays.

2. Select the family to update under the **Copy Similar Families** hierarchy.

3. Click **Synchronize**.

   The Copy by Family ribbon displays.

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

   > TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group during the original copy. For more information, see *Copy Objects in a Similar Family* (on page 319).

6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**. Optionally, select **Multiple** to copy to more than one target group.

7. Click **Find and correlate objects** .

   The software highlights all existing structural objects that are similar, meet the criteria of both the Source Group and the Target Group, and have been added after the original

copy.

8. Click **Modify target group copy** on the ribbon.

   *The software highlights all the objects that are in the **Source Group** but not in the **Target Group**, in both the **Source Group** and the **Target Group** but have changed, or have been added after the original copy.*

9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to green.

10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

📝 **NOTE** Within the **Create Target Group** and **Modify Target Group**, you can manually select objects to copy. See *Manually Copy Objects* (on page 324).

Delete objects in a similar family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to work with under the **Copy Similar Families** hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Group** box, select the group in which objects have been deleted. By default, this is the group indicated as the source group during the original copy. For more information, see *Copy objects in a similar family* (on page 319).

5. In the **Target Group** box, select the group in which to remove the deleted objects from the **Source Group**. Optionally, select **Multiple** to copy to more than one target group.

6. Click **Delete target group copy** on the ribbon.

   *The software highlights all the objects in the **Target Group** that have been deleted from the **Source Group**.*

7. Optionally, select any highlighted objects in the **Target** group that you do not want deleted.

   The color of the highlighted objects changes from yellow to green.

8. Click **Finish**.

   *The software deletes the highlighted objects in the **Target** group. Objects highlighted in green are* not *deleted from the **Target Group**. The software marks the green objects as recognized differences between the two groups and excludes the objects from future copy operations.*

Copy objects in a symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to work with under the 🛡️ **Copy Symmetry Families** hierarchy.

   💡**TIP** For instructions on creating a new copy symmetry family, see *Create a copy symmetry family* (on page 316).

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

   💡**TIP** The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group when you defined the family. For more information, see *Symmetry Family Properties Dialog Box* (on page 333).

6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**.

7. Click **Find and correlate objects** 🔍.

   The software highlights all existing structural objects that have symmetry.

8. Click **Create target group copy** 🛡️ on the ribbon.

   *The software highlights in yellow all the objects in the **Source Group** that are not in the **Target Group**.*

9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to cyan.

10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

Update objects in a symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to update under the **Copy Symmetry Families** hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the filter that the copy uses to find structural objects. By default, **Workspace Filter** is used.

   TIP The **<Filter> + Workspace Filter** option allows you to select a predefined filter from the **Select Filter** dialog box so that the operation is performed on only those objects satisfying both the predefined filter and the Workspace filter.

5. In the **Source Group** box, select the group from which to copy objects. By default, this is the group indicated as the source group during the original copy. For more information, see *Symmetry Family Properties Dialog Box* (on page 333).

6. In the **Target Group** box, select the group to which to copy objects from the **Source Group**.

7. Click **Find and correlate objects** .

   *The software highlights all existing structural objects that are similar, meet the criteria of both the **Source Group** and the **Target Group**, and have been added after the original copy.*

8. Click **Modify target group copy** on the ribbon.

   *The software highlights all the objects that are in the **Source Group** but not in the **Target Group**, in both the **Source Group** and the **Target Group** but have changed, or have been added after the original copy.*

9. Optionally, select any highlighted objects in the graphics view that you do not want to copy to the **Target Group**.

   The color of the highlighted objects changes from yellow to green.

10. Click **Finish**.

    *The software copies the highlighted objects in the **Source** group to the **Target** group.*

NOTE Within the **Create Target Group** and **Modify Target Group**, you can manually select objects to copy. See *Manually copy objects* (on page 324).

Delete objects in a symmetry family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the family to work with under the 🛡️ **Copy Symmetry Families** hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Group** box, select the group in which objects have been deleted. By default, this is the group indicated as the source group during the original copy. For more information, see *Symmetry Family Properties Dialog Box* (on page 333).

5. In the **Target Group** box, select the group in which to remove the deleted objects from the **Source Group**.

6. Click **Delete target group copy** 🛡️ on the ribbon.

   *The software highlights all the objects in the **Target Group** that have been deleted from the **Source Group**.*

7. Optionally, select any highlighted objects in the **Target** group that you do not want deleted.

   The color of the highlighted objects changes from yellow to green.

8. Click **Finish**.

   *The software deletes the highlighted objects in the **Target** group. Objects highlighted in green are* not *deleted from the **Target Group**. The software marks the green objects as recognized differences between the two groups and excludes the objects from future copy operations.*

Manually copy objects

When using the **Create target group copy** 🛡️ and **Modify target group copy** 🛡️ commands on the **Copy by Family** ribbon, you can manually select items to copy. Select from the following two methods:

Copy Preselected Objects

1. Pre-select the needed objects.

2. Click **Edit** > **Copy by Family**.

3. To create a target, select **Create target group copy** 🛡️, or click **Finish**.

   Preselected objects are copied. Objects on which the selected objects are dependent are also copied.

4. If you want to modify an existing target, select **Modify target group copy** 🛡️, or click **Finish**.

   Preselected objects are copied. Objects on which the selected objects are dependent are also copied.

📝 NOTE Only the target group is modified, even if you select target and source objects.

Clear Objects and Select Again

1. Click **Edit** > **Copy by Family.**

2. Clear the **Select All** checkbox, and select **Create target group copy** .

    All the copy candidates are highlighted in green, indicating that they are no longer selected.

3. Manually select the objects to copy.

Copy objects using Copy Shortcuts

You can replicate objects under similar or symmetry families and update the replicas using customized shortcut keys.

Configure Copy Shortcuts

1. Define the shortcut keys in the CopybyFamilyShortCutKeys.txt file. The file is located in the [*Product Folder*]\CopybyFamily\Data folder.

2. Click **Edit** > **Copy by Family**.

3. In the **Select Copy Family** dialog box, click **Copy Shortcuts**.

    *The **Copy Shortcuts Settings** dialog box displays.*

4. Configure each shortcut key as follows:

    a. In the **Family** cell, specify **Similar** or **Symmetry**.

    b. In the **Source Group** cell, specify the family group to use as the copy source.

    c. In the **Target Group** cell, specify the family group to create or modify.

    d. In the **Select Child** cell, specify **True** to include child objects in the copy operation, or **False** to exclude them. The default value comes from the **Copy by Family** ribbon.

    e. In the **Molded Form Only** cell, specify **True** if the shortcut key is used only in the **Molded Forms** task, or **False** if it is used across tasks. The default value comes from the **Copy by Family** ribbon.

5. In **Log File Path**, specify a log file for storing the copy shortcuts operation.

6. Click **OK**.

Replicate Objects Using Shortcut Keys

1. In the **Workspace Explorer**, select the objects under the family range to be replicated or to update their replicas.

2. Press the assigned copy shortcut key to start the copy shortcuts operation.

    A dialog box displays showing the corresponding family name and stating that the copy or modify operation is in progress. When the operation is completed, the dialog box closes automatically.

NOTES

▪ If any error occurs during the copy shortcuts operation, a dialog box displays allowing you to display the log file.

- If there are any unresolved entities on the target side, the browser automatically displays these entities so that you can resolve them manually.

Copy objects in an across model family

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Click **Create New Across Model Family** , and create a copy across model family. For information on how to create a across model family, see *Create a copy across models family using a single grid plane* (on page 317) and *Create a copy across models family using multiple grid planes* (on page 318).

3. Select the **Copy Across Model Family** from the **Copy Across Families** hierarchy.

4. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

5. Click **Create target group copy** .

6. Click **Finish** to copy the objects from the source ship to the target ship.

Find and correlate objects across models

1. In the target ship, click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Click **Create New Across Model Family** , and create a copy across model family. For information on how to create a across model family, see *Create a copy across models family using a single grid plane* (on page 317) and *Create a copy across models family using multiple grid planes* (on page 318).

3. Select the **Copy Across Model Family** from the **Copy Across Families** hierarchy.

4. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

5. Click **Find and correlate** .

   The objects which can be mapped are identified.

6. Click **Finish** to correlate the objects to the target ship from the specified source ship.

   NOTE After you perform **Find and correlate**, the software includes the objects to the copy family so that duplicates are not created during the synchronize operation.

Select the source filter for copy across models

You can perform copy across models without selecting a filter (No Filter) or by specifying the Source Filter.

Performing Copy Across Models without specifying a Source Filter

1. In the target ship, click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the **Copy Across Model Family** from the **Copy Across Families** 🛡 hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays.*

4. In the **Source Filter** box, select the **No Filter** option.

5. Click **Create target group copy** 🛡.

   *The message **As Source Filter is not specified, all the entities in Source Ship will be processed. This could take considerable time. Please press OK to continue or Cancel to stop the operation.***

6. Click **Yes** to perform the copy or **No** to cancel the copy and specify a different filter.

Performing Copy Across Models specifying a Source Filter

1. In the target ship, click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Select the **Copy Across Model Family** from the **Copy Across Families** 🛡 hierarchy.

3. Click **Synchronize**.

   *The **Copy by Family** ribbon displays. The **Source Filter** list shows your previously used filters and the **No Filter** and **Select Filter** options. By default, your last used filter is selected. If this is the first time you synchronize the family, **No Filter** is the default.*

4. In the **Source Filter** box, select a filter. To select a filter corresponding to those in the source model of the selected copy across model family, select **Select Filter**. Select a filter from the **Select Filter** dialog box, and click **OK**.

   💡 TIP The **Source Filter** box does not allow you to type a filter name.

5. Click **Create target group copy** 🛡.

   If the filter exists, the copy candidates are selected based on the objects associated with the filter.

   💡 TIP If the specified filter does not exist, the message **Filter with the given name does not exist in Source Ship. Please enter the name of an existing Filter** displays. Select **Source Filter for Copy Across Models**.

Schedule a Copy by Family operation as a batch job

1. Click **Edit > Copy by Family**.

   *The **Select Copy Family** dialog box displays.*

2. Click **Batch**.

   *The **Copy by Family Batch** dialog box displays.*

   ♦ TIP You must configure the batch queues before you can submit a Copy by Family operation as a batch job. For more information, see *Configure the Smart 3D batch queues* (on page 328).

3. Set up the parameters to meet your needs. For more information, see *Copy By Family Batch Dialog Box* (on page 339).

4. Click **Submit**.

   *The **Schedule [Task]** dialog box displays.*

5. Set up the scheduling parameters to meet your needs. For more information, see *Schedule [Task] Dialog Box* (on page 340).

Configure the Smart 3D batch queues

1. Run the configureSP3DBatchQueue.exe utility. By default, this utility is downloaded to the *[Product Folder]*\ProjectMgmt\Tools\Bin folder.
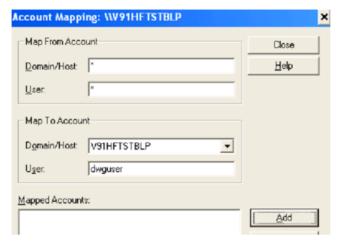
   The software creates, enables, and starts the SP3DProjectManagement, SP3DPrinting, and SP3DApplication queues.

Configure account mapping for batch services

1. Click **Start** > **Intergraph Batch Manager**.

2. Click **Option Server** > **Connect To** to connect to individual servers.

3. Right-click a server, and select **Account Mapping**.

   *The **Account Mapping** dialog box displays.*

4. Type the account mapping details. In the following example, all users and domains from **Map From Account** are mapped to the **Domain/Host: V91HFTSTBLP** and **User: dwguser** account.

5. Click **Add**.

6. Type the password to complete the mapping.

NOTE Other mapping options are also possible. For example, the jobs submitted by User-1 can be mapped to Account-1 while the jobs submitted by User-2 can be mapped to Account-2.

Synchronize Options Dialog Box

Find Options

Clear relationships for Source Filter before Find

Removes any relationships created in the family after clicking **Find and correlate objects** . This restores highlighting of objects in the graphics view for the **Find and correlate objects** step. **Clear then Find** performs the same function as **Refresh Family** on the **Select Copy Family** dialog box.

CAUTION This option removes all **Copy By Family** relationships in the current family for the objects in the source filter.

To In/Outboard

Changes all port and starboard molded conventions to inboard and outboard before using the **Find and correlate objects** command.

Manual Mode

Manually find objects that were not found by the software when using the **Find and correlate objects** command. You can select an object for the **Source Group** and an object for the **Target Group** to make them similar or symmetrical.

Create/Modify Options

Molded Forms objects only

Selects molded forms objects.

Select All

Selects all of the objects.

Select Child

Selects the child objects.

Modify Only Options

Geometry Modifications
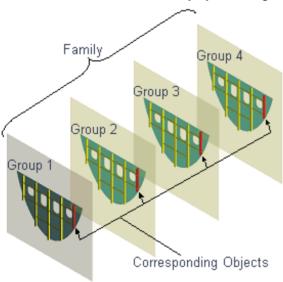
Finds changes in landing curves.

Similar Family Properties Dialog Box
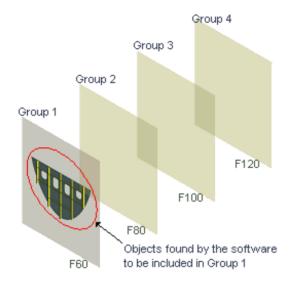
Specifies the groups and grid planes for a Copy Similar family.

Overview

The Copy Similar method uses a user-defined family of similar grid planes to define what structural objects are copied and to maintain the relationships between the objects to check for modifications. You define a family by selecting the grid planes for each group in the family.

In the following example, Group1 is the source group. The software finds all plate systems that use the Group1 transverse grid plane as the surface definition. The software then finds all children of the plate system (seams, stiffeners, openings, and features). Finally, the software finds all the objects that are bounded by either the select grid plane or the plate systems defined by the grid plane and all their children.

Group 4

Group 3

Group 2

Group 1

F120

F100

F80

Objects found by the software
to be included in Group 1

F60
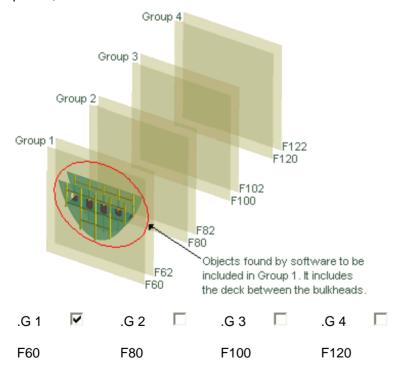
You can use one grid plane or multiple grid planes in each group to define the family.

In this example, one grid plane defines each group, with Group1 being the source group to be cloned for the family.

.G 1 ☑   .G 2 ☐   .G 3 ☐   .G 4 ☐

F60   F80   F100   F120

In the next example, two transverse grid planes define each group, with Group1 being the source group to be cloned for the family. Objects between the grid planes, but related to the grid planes, are also included.

Group 4

Group 3

Group 2

Group 1

F122
F120

F102
F100

F82
F80

Objects found by software to be
included in Group 1. It includes
the deck between the bulkheads.

F62
F60

.G 1 ☑   .G 2 ☐   .G 3 ☐   .G 4 ☐

F60   F80   F100   F120

| F62 | F82 | F102 | F122 |

Dialog Box Controls

Name

Type a name for the family. This name must be unique among all family names.

Standard

Defines the Copy Similar Family by specifying one or more rows of grid planes to be used for the groups. For example:

.G 1 ☑	.G 2 ☐	.G 3 ☐	.G 4 ☐
F60	F80	F100	F120
F61	F81	F101	F121
F62	F82	F102	F122
F63	F83	F103	F123
F64	F84	F104	F124

By Range

Defines the Copy Similar Family by specifying a range of grid planes. All grid planes within a range are included. In this example, the results are the same as the five rows specified for the **Standard** example:

.G 1 ☑	.G 2 ☐	.G 3 ☐	.G 4 ☐
F60	F80	F100	F120
F64	F84	F104	F124

Overlapping

Defines the Copy Similar Family by specifying a range of grid planes, but ignores structure that overlaps from one group to the next. For example, in the both the **Standard** and **By Range** examples, a transverse bulkhead could be at the end of the F60-F61 range, but could also be at the beginning of the F61-F62 range. The **Overlapping** option prevents the F61 bulkhead from being created twice.

.G X

Displays the groups that you have defined for this family. Check the corresponding box for the group that you want to copy to the other groups. To add a group to the family, click **Insert group**. To remove a group from the family, click **Remove group**. The name of the group can be modified. For example, **.G 1** can be changed to **Group 1** or **F60 - 62 Group**.

Insert group

Adds a new group to the family.

Remove group

Removes the selected group from the family.

Insert corresponding item (row)

Inserts another row. Click this button when you want to define a group using multiple grid planes.

See Also

Symmetry Family Properties Dialog Box

Specifies the groups and symmetry planes for a Copy Symmetry family.

For each symmetry family, a coordinate system is selected, typically the main global coordinate system. The Y=0 grid plane, the longitudinal centerline of the coordinate system, is automatically used by the software as the symmetry plane. The software automatically creates two symmetrical groups, one on each side (port and starboard) of the symmetry plane, containing all the grid planes for each side. One group is selected as the source group to be cloned. For example, the software uses all port side grid planes as the source group. The starboard side grid planes are the corresponding elements for the starboard group. To determine all the structural objects that belong to the source group, the software finds all objects whose definition is based on the grid planes, then finds all their children, then finds all the objects that are bounded by either the grid planes or the systems defined by the grid planes and all their children.

Main tab

Name

Type a name for the copy symmetry family. This name must be unique among all family names.

Symmetry Planes

Select the coordinate systems to use for the symmetry plane. By default, the software uses the Y=0 plane of the selected coordinate system as the symmetry plane. If you select more than one coordinate system, all selected coordinate systems must have coincident Y=0 planes.

Source Group

Select which group is the source for the symmetrical copy.

Port Group Name

Type a name for the port symmetrical group. This name appears when you synchronize the groups. The default name is **Port**.

Starboard Group Name

Type a name for the starboard symmetrical group. This name appears when you synchronize the groups. The default name is **Starboard**.

Range Growth Value

Type a value for the range required for selecting the Copy candidates.

Copy MfgData

Copies manufacturing objects.

Configuration tab

Displays the creation, modification, and status information about an object.

Ship

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which object belongs. You can select another permission group, if needed. Permission groups are created in the Project Management task.

Status

Specifies the current status of the selected object or filter. Depending on your access levels, you may not be able to change the status of the object.

Created

Displays the date and time that the object was created.

Created by

Displays the user name of the person who modified the object.

Modified

Displays the date and time when the object was modified.

Modified by

Displays the user name of the person who modified the object.

See Also

Across Model Family Definition Dialog Box

Defines an Across Model family. An Across Model family is used to perform Copy Similar across ships.

★ IMPORTANT To use this functionality when your target and source ship are on different servers, you must link the servers. For more information, see *Linking Servers* in the *Intergraph Smart*TM *3D Installation Guide*.

Click **Create New Across Model Family** 🛡 to open the dialog box.

Main Tab

Name

Type a name for the across model family. This name must be unique among all family names. The name **New Across Model Family** is available by default.

Range Growth Value

Type a value for the range required when selecting the Copy candidates. The **Range Growth Value i**s set to **0.80** m by default.

Copy MfgData

Copies manufacturing objects.

Method

Select the method of the definition of the family. You can select either by **Range** or **Standard**. The **Standard** method is selected by default.

Ship Name

Displays the name of the target or source ship. The target ship name (name of the current/active ship) cannot be edited. You can select ships within a site, even if they belong to different servers.

Group Name

Specifies the name of the target/source group. By default, the name of target/source group is the same as the target/source ship name. The target/source group name can be edited.

Corresponding Elements 1

Type a frame name, or browse and select from the list in the **Select Frame** dialog box.

📝 NOTE If the frame name you type is not valid or not available, it is highlighted in red. If you type a new corresponding element name, use the following format:

Coordinate System Name:Frame name. For example, *CS_0:F0* where CS_0 is the Coordinate System name and F0 is the frame name.

Add Corresponding Element

Adds a row for you to type or select corresponding elements.

📝 NOTE Because there are only two corresponding elements allowed when the method definition is **By Range**, the **Add corresponding Elements** button is not available.

Delete Corresponding Element

Removes an element that you have added. This command is available only after you add

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corresponding elements.

NOTE When the method definition is **By Range**, the **Delete corresponding Elements** button is not available.

See Also

Select Filter Dialog Box

Creates, edits, deletes, and selects filters for use with the **Define Workspace**, **Surface Style Rules**, and other **Select by Filter** commands, including Project Management's **Model Data Reuse** (MDR), Drawings View Styles, and Reports commands that require runtime filter selection. You can access this dialog box in several ways.

- Select **File** > **Define Workspace**, and select the **More** option in the **Filter** box.

- Select **Format** > **Surface Style Rules**, click **New** or **Modify**, and then select the **More** option in the **Filter** box.

- Select **Tools** > **Select by Filter.**

The tree view displays the following types of filters:

- **Catalog Filters** - These filters are used to reference data in the Catalog. For example, a catalog filter could apply to company-wide operations. Your administrator can define Company_Filter_1, Company_Filter_2, and so forth.

- **Model Filters** - These filters are available to everyone assigned to a specific model database. There are delivered catalog filters to query on the different types of model objects. You must have the appropriate privileges to create, edit, or delete these filters.

- **My Filters** - These are personal filters that you create and place in the **My Filters** folder. They are visible only to you, the owner. You cannot see the personal filters of others, and they cannot see your personal filters. Select a filter from one of the listed filters, or create a new filter to meet your specific requirements.

New Folder

  Creates a new folder.

New Filter (Simple or Asking)

  Displays the **New Filter Properties** dialog box so that you can create a new filter. Asking filters allow you to specify the parameters of the search. An asking filter has built-in functionality to ask for values (with boxes that you are required to supply). The values apply to properties that you have already designated you will supply when the filter runs. Asking filters are portable between models.

  NOTE Model Data Reuse (MDR) does not support asking filters. The only valid filter types for an MDR transaction are System, Permission Group, Object Type, Volume and Properties. You can define the filter on any one of these tabs or in a combination using multiple tabs.

New Compound Filter

  Displays the **New Compound Filter Properties** dialog box, which you use to create a new

compound filter containing the Or, And, or Not operators. Compound filters are not supported for MDR.

New SQL Filter

Displays the **New SQL Filter Properties** dialog box, in which you can type the text of an SQL query. SQL filters are not supported for MDR

Delete

Removes a filter or folder from the **Select Filter** list. If you delete a folder, the software also deletes its contents.

Rename

Changes the name of an existing filter or folder from the **Select Filter** list.

Properties

Displays the **Filter Properties** dialog box so that you can select the properties that determine your filter search criteria.

NOTES

- If this dialog box is activated from the **Select by Filter** command, you can select multiple filters on this dialog box. Hold CTRL or SHIFT, and click each filter. When you click **OK**, all objects that fit the selected filters are selected.

- If this dialog box is activated from the **Select by Filter** command, it clears the select set before adding objects to the select set.

Select Copy Family Dialog Box

Toolbar

Create New Similar Family

Creates a new copy similar family. For more information on creating copy similar families, see *Similar Family Properties Dialog Box* (on page 330).

Create New Symmetry Family

Creates a new symmetry family. For more information on creating copy symmetry families, see *Symmetry Family Properties Dialog Box* (on page 333).

Create New Across Model Family

Creates a new across model family. For more information on creating across model families, see *Across Model Family Definition Dialog Box* (on page 335).

Delete Family Definition

Deletes the selected family.

Rename Family Definition

Renames the selected family. You must select a family before you can rename that family.

Edit Family Definition

Edits the selected family. You must select a family before you can edit that family.

Refresh Family

Removes any relationships created in the family by using the **Synchronize** command and the **Copy by Family** ribbon. For example, **Refresh Family** restores highlighting of objects in the graphics view for the **Find and correlate objects** step if that step has been previously carried out.

Copy Shortcuts

Configures copy shortcut keys for similar or symmetry families and specify a log file to store the copy operation. For more information, see *Copy Shortcuts Settings Dialog Box* (on page 338).

Copy Family Hierarchy

Copy Across Model Families

Lists the Copy Across Model Families that you have defined.

Copy Similar Families

Lists the copy similar families that you have defined.

Copy Symmetry Families

Lists the copy symmetry families that you have defined.

Batch

Displays the **Copy by Family Batch** dialog box. You can use this dialog box to execute jobs in a batch queue. For more information, see *Copy By Family Batch Dialog Box* (on page 339).

Synchronize

Opens the **Copy by Family** ribbon for the selected family.

Close

Closes the dialog box.

Copy Shortcuts Settings Dialog Box

Configures shortcut keys for similar or symmetry families, and specifies a log file for the copy shortcuts operation.

SNo

Displays the number assigned to the shortcut key.

ShortCut Key

Displays the shortcut key defined in the CopybyFamilyShortCutKeys.txt file. This text file is located in the *[Product Folder]*\CopybyFamily\Data folder.

Family

Indicates whether to copy similar families or symmetry families.

Source Group

Specifies the family group to use as the copy source.

Target Group

Specifies the family group to create or modify.

Select Child

Determines whether to include child objects in the copy operation. Specify **True** or **False** for the value. The default value comes from the **Copy by Family** ribbon.

Molded Form Only

Determines whether the shortcut key applies only in the **Molded Forms** task. Specify **True** or **False** for the value. The default value comes from the **Copy by Family** ribbon.

Log File Path

Specifies a log file for storing the last copy shortcuts operation.

Browse

Click to browse to the log file.

Open

Opens the log file.

OK

Accepts the configuration.

Cancel

Rejects the configuration and closes the dialog box.

See Also
Copy by Family (on page 309)
Copy By Family Log Files (on page 342)
Copy objects using Copy Shortcuts (on page 325)
Select Copy Family Dialog Box (on page 337)

Copy By Family Batch Dialog Box

Controls properties for copy by family batch processes.

Actions To Queue

Displays the actions in the queue. The options in the grid behave in the same way as those on the Copy by Family ribbon. For more information on these options, see *Copy by Family* (on page 309).

Family

Specifies the family on which to perform the batch operation. After you select a value for **Family**, the software invokes the source and target groups and source filter.

Steps

Specifies the operation to perform. The list contains the following options:

- 🔍 **Find**
- 🛡 **Create**

- **Modify** - This option is not available for copy across model.

- **Delete** - This option is not available for copy across model.

- **Object Report**

- **Property Report**

Source Group

Displays the source group. The software determines this based on the value in the **Family** box.

Target Groups

Displays the target groups. The software determines this based on the value in the **Family** box.

Source Filter

Displays the source filter. The software determines this based on the value in the **Family** box.

Properties

Displays the **Properties** dialog box. For more information, see *Properties Dialog Box* (on page 342). This option is not available if the **Steps** box is set to **Object Report** or **Property Report**.

Add Action

Adds a row to the end of the grid.

Delete Action

Removes the selected row from the grid. This is the same as pressing DELETE.

Save CBF Log File

Displays the location for the Copy by Family log file. You can type the path, or click **Browse** to navigate to the folder.

Audit Report Path

Displays the location for the Copy by Family audit report file. You can type the path, or click **Browse** to navigate to the folder.

Submit

Displays the **Schedule [Task]** dialog box. For more information see *Schedule [Task] Dialog Box* (on page 340)

Schedule [Task] Dialog Box

Queue

Displays the name of the queues configured by an administrator for the job. For more information on configuring the queues, see *Configure Queues for Jobs* in the *Project Management User's Guide*.

Run job

Sets the frequency with which the job runs. Jobs can be scheduled to run once or on a

regular interval (daily, weekly, or monthly). Depending on the job frequency selected, additional controls display. These controls allow you to define more specific scheduling information. The scheduling controls can be changed only at job submission.

Run on

Sets the time to start running the job.

Options

Opens the Optional Schedule Properties Dialog Box that you can use to define a start and end date.

Run on box

Contains a calendar from which you can select the run date. This option is available when you select **Once** from **Run job**.

Every X days

Specifies how many days pass between job runs. This option is available when you select **Daily** from **Run job**.

Every X weeks

Specifies how many weeks pass between job runs. In addition, you can select on which days the job runs. This option is available when you select **Weekly** from **Run job**.

Day X of the month

Specifies on which day of the month the job runs. This option is available when you select **Monthly** from **Run job**.

The X Y of the month

Specifies on which day of the month the job runs. For example, you can select the last Monday of the month. This option is available when you select **Monthly** from **Run job**.

Job Start

Notifies you when the job starts, if Outlook is set up.

Job Completion

Notifies when the job completes, if Outlook is set up.

Job Abort

Notifies you if the job aborts, if Outlook is set up.

Address Book

Selects the name of the person to be notified by e-mail of the job status, if Outlook is set up. If Outlook is not available, this option does not work. You can also type the address manually. The person you define here receives an email with the job log files after the job finishes.

NOTES

- The Batch Services SMTP option must be configured on the batch server for this to work. For more information, see the Intergraph Smart Batch Services documentation.

- The WinZip application is no longer required on the batch server to compress any emailed attachments. Compression is now done with functionality included in Smart 3D.

Properties Dialog Box

Controls properties for Copy by Family batch operations.

Clear relationships for Source Filter before Find

Removes all Copy by Family relationships in the current family for the object in the source filter.

Molded Form Objects Only

Indicates that only Molded Forms objects are used for create and modify batch operations.

Select All

Indicates that all objects are used for create and modify batch operations.

Select Child

Indicates that child objects are used for create and modify batch operations.

Geometry Modifications

Indicates that geometry modifications are included in modify batch operations.

Copy By Family Log Files

Logs are generated for **Find and correlate objects** , **Create target group copy** , **Modify target group copy** , and **Finish** operations.

Find and correlate objects log

The purpose of this log is to show objects that failed to find operations because the target area contains duplicated objects (both geometrically and topologically). You must delete one of the duplicates before the **Find and correlate objects** command can correlate the source and target objects and any dependent objects.

NOTE If the **Modify Port/Starboard** option is selected, the list of forced objects is logged at the finish of the **Find and correlate objects** command.

Create target group copy log

The initial step of this command shows the candidates for copy (that is, objects that do not yet have a target object). Any objects that are on the to-do list are not selected for copy. None of the dependents of the to-do list objects are selected for copy.

The list of objects on the to-do list is logged, and any dependent objects are listed at the end.

Modify target group copy log

Modify target group copy excludes to-do list objects and reports on them in the same way as the **Create target group copy** command does. **Modify target group copy** compares the attribution of the source object with the target object and selects candidates to copy when differences are found. To simplify the selection process, these differences are reported in the log.

The actual attribute difference is shown first with the objects that are different shown at the end.

Because features are considered attributes of parts, this creates a hierarchy. All features, including intermediate features, are shown.

Finish after Create, Modify, or Delete log

Errors in the finish log include:

- New errors, such as targets that go on the to-do list

- Exceptions thrown by the underlying operations such as copy, split, or detail.

- Exceptions thrown by **Copy by Family** because it is processing an object before its parent. This requires that the copy be run a second time.

The format of these error messages is as follows:

1	!Molded Form target object in error A-614DCK->A-2602DCK / Plate
	!See Copy Error Management Workflow in Copy By Family User Aid.
	!Split or Part target object in error A-614DCK_PART->A-2602DCK_PART / Part
	!See Copy Error Management Workflow in Copy By Family User Aid.
2	Copying: H170P-B27BP-1P -type: Solid
	!!!Exception processing above elements
	!!!These elements fail to process, repair and rerun
	!!!Report error to Intergraph
3	!Skipped D51(2nd Dk)-491DCK / Plate
	!Note: Above elements have been skipped, run copy a second time

If the target geometry is bad, the dependents are copied unless they cannot be copied without the parent. The dependents that are not copied are listed at the end of the log.

See Also

Copy by Family (on page 309)
Select Copy Family Dialog Box (on page 337)
Copy By Family Log File Workflows (on page 343)
Similar Family Properties Dialog Box (on page 330)
Symmetry Family Properties Dialog Box (on page 333)

Copy By Family Log File Workflows

Target is on the To-Do List

If the **"!Molded Form target object in error A-614DCK->A- 2602DCK/Plate"** type of error is the first error in the log file, check the unsupported object construction list.

- If the source object's construction is not supported by copy, try to fix the target object manually or recreate it manually on the target side.

- If the source object's construction is supported by copy, select the **Clear Before** check box, run **Find and correlate objects**, and try copying again.

After the target object is repaired or created, select the **Clear Before** check box, run **Find and correlate objects** and **Create/Modify/Delete** again to copy any remaining objects.

If the **"!Split or Part target object in error A- 614DCK_PART->A-2602DCK_PART/Part"** type of error is the first error in the log file, check the unsupported object construction list.

- If the source object's construction is not supported by copy, try to fix the target object manually by undetailing and redetailing the parts or by fixing the molded forms system and boundaries. Alternately, you can recreate the object manually on the target side.

- If the source object is supported by copy, contact Intergraph Support, and model the failed target object manually.

After the target object is repaired or created, select the **Clear Before** check box, run **Find and correlate objects** and **Create/Modify/Delete** again to copy any remaining objects.

Exception

If the **"!!!Exception processing above elements"** type of error is the first error in the log file, do the following:

- If the failed object is a Molded Forms object, select the **Clear Before** checkbox, run **Find and correlate objects** again, and then try copying again.

- If the failed object is a detailed part, or if the second try to copy is unsuccessful, contact Intergraph Support, and model the failed target object manually. After the object is repaired or created, run **Find and correlate objects** and **Create**/**Modify/Delete** again to copy any remaining objects.

Object Skipped

If the **"!Skipped D51(2nd Dk)-491DCK/Plate"** type of error is found in the log file, then there was a problem with the order in which the objects were copied. If you run copy again, the objects are copied correctly, and errors are eliminated from the log.

If errors remain, then there was a problem with the parent of the failed object specified in the skipped error log. Fix the parent so that this object can be copied successfully.

Entities not Completely Supported by the Copy Process

Limitations of the copy implementation can be classified in the following types:

- Object types not yet supported by copy similar/symmetry. The objects are excluded from the copy process. That is, they are not selected as candidates. In some cases, dependent entities might be selected for copy and could fail to copy. These specific cases are listed below.

- Some aspect of the construction of the object is not supported. In this case, you must edit the object to remove the unsupported aspect. These specific constructions are listed below.

Object Types Not Supported By Copy Similar/Symmetry

- Auxiliary coordinate systems. Dependent objects fail.

- Manual logical connections.

- Inverted connections.

- Beam parts (detailed property and beams using assoc points on edges).

- Standalone Plate parts, Profile Parts, Lapped Plate Parts, Bracket Parts.

- Standalone Profile Parts, Profile Edge Reinforcement Parts.

- Straking Seams.

- Profile Straking Seams.
- Split Physical Connections.
- Manufacturing Plates and Profiles.

Aspects of the Construction of Object Not Supported

- Problem with profile and seams by table.
- RAD Sketches containing constraints. After copy, the software replaces the inputs to the sketcher with the new copied geometry. If the new geometry is not infinite, and if the direction of the curves is reversed, this produces bad results for end point constraints and any constraints dependent on the direction. **Copy Symmetry** with edge constraints is not supported.
- Assoc point on edge. Using the place point on edge **Smart Point** placement can result in the point reversal during mirror.

See Also
Copy By Family Log Files (on page 342)

Frame mapping and Range Growth Values in Copy by Family
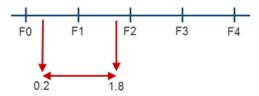
The **Range Growth Value** and **Offset** values determine the range of the family from which you must pick the copy candidates. The technique for using **Range Growth Value** depends on the type of family with which you are working.

Similar Standard Family

If you create a similar standard family with F1 as the source and F4 as the target, the software maps F1 to F4.
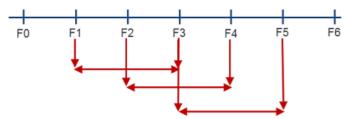
If you specify a default **Range Growth Value** of 0.8, then the range of the family extends from the source frame -0.8 to the source frame + 0.8
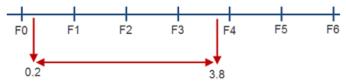
Similar Overlapping Family

To create a similar overlapping family you must have at least one frame in common between the source and the target. For example, you could create a family with F1 and F3 as the source and

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F3 and F5 as the target. In this case, F3 is the common frame. The software then maps F1 to F3, F2 to F4, and F3 to F5.
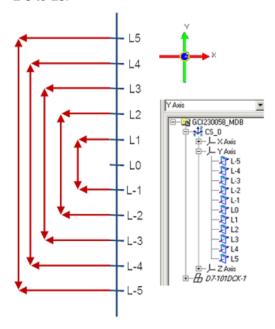
If you specify a default **Range Growth Value** of 0.8, Smart 3D sets the range of the family from the minimum source frame value - 0.8 to the maximum source frame value + 0.8
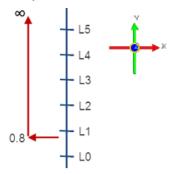
Symmetry Family

If you create symmetry with port as the source with five positive frames and starboard as the target with five negative frames, Smart 3D maps L-1 to L1, L2 to L-2, L -3 to L3, L-4 to L4, and L-5 to L5.
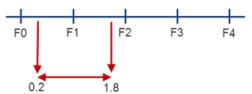
If you specify a default **Range Growth Value** of 0.8, Smart 3D sets the range of the symmetry family from the centerline + 0.8 to infinity.

Across Models Standard Family

If you create an Across-Models Standard Family with F1 as the source and F4 as the target, Smart 3D maps F1 to F4. This behavior is the same as creating a standard family within a model. The software also maps the X and Y axes in an Across Models Family. If you specify a default Range Growth Value of 0.8, Smart 3D sets the range of the family from the source minimum frame value - 0.8 to the source maximum frame value + 0.8.

Objects and behaviors supported for Copy by Family

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
Molded Forms	Planar Plate ▪ Using plane created by **Plane by Vector Normal** ▪ Using plane created by **Plane By Point Normal** ▪ Using plane created by **Plane By 3 Points**	Yes	Yes	Yes
	LEP	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	NLEP	Yes	Yes	Yes
	Advanced Plate System ▪ APS polyline plate macros ▪ APS corrugated plate macros ▪ APS swage plate macros	Yes	Yes	Yes
	Bracket by Plane ▪ Using plane created by coincident plane ▪ Using plane created by **Plane By Vector Normal** ▪ Using plane created by **Plane By 3 Points**	Yes	Yes	Yes
	Bracket by selected supports	Yes	Yes	Yes
	Profile by intersection	Yes	Yes	Yes
	Profile by projection	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Profile by geometric construction	Yes	Yes	Yes
	Profile by offset	Yes	Yes	Yes
	Profile by table	Yes	Yes	Yes
	Tripping stiffener	Yes	Yes	Yes
	Profile edge reinforcement	Yes	Yes	Yes
	Seam by intersection	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Seam by offset	Yes	Yes	Yes
	Seam by table	Yes	Yes	Yes
	Seam by geometric construction	Yes	Yes	Yes
	Profile seam	Yes	Yes	Yes
	Openings	Yes	Yes	Yes
	Macro or Catalog openings	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Standard Members	Yes	Yes	Yes
	Built-up (Designed) Members	Yes	Yes	Yes
	Generated logical connections	Yes	Yes	Yes
	Split logical connection having inverted regions	Yes	Yes	Yes
	Manual logical connection	Yes	Yes	Yes
	Plate knuckles	Yes	Yes	Yes

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Profile knuckles	Yes	Yes	Yes

NOTE **Copy by Family** also supports brackets created with **Tripping Bracket**.

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
Structural Detailing	Standalone plate part	Yes	Yes	No
	Lapped plate part	Yes	Yes	No
	Plate part	Yes	Yes	No
	Bracket part	Yes	Yes	No
	Profile part	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Edge features	Yes	Yes	No
	Corner features	Yes	Yes	No
	Sketch features	Yes	Yes	No
	Edge treatments	Yes	Yes	No
	Manual chamfer under edge reinforcement	Yes	Yes	No
	Generated chamfer	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
	Smart plate	Yes	Yes	No
	Assembly connections	Yes	Yes	No
	Physical connections	Yes	Yes	No
	End cuts	Yes	Yes	No
	Slot	Yes	Yes	No
	Collar	Yes	Yes	No

Task	Objects Supported for Copy in Copy by Family	Similar Family	Symmetry Family	Copy Across Model Family
Structural Manufacturing	Constant margin	Yes	Yes	No
	Oblique margin	Yes	Yes	No
	Marking	Yes	Yes	No
	Shrinkage	Yes	Yes	No

Objects not supported by Copy by Family

Task	Objects Not Supported by Copy by Family
Molded Forms	Objects created using **Reference Curve by Intersection**
	Ruled plates
	Child plates
Structural Detailing	Bracket part
	Standalone stiffener part
	Beam profile part
	Plate edge reinforcement part
	Profile edge reinforcement part
	Profile straking seam
	Split physical connection
	Member generic assembly connection
	Manual physical connection
Structural Manufacturing	Template
	Pin Jig
	Manufacturing Plate
	Manufacturing Profile
⊨ NOTE No other objects from other tasks (such as Equipment, Hangers and Supports, and so on) are supported in **Copy by Family**.	

Objects and behaviors supported in Attribute Modify

Task	Objects Supported for Modify in Copy by Family	Properties Supported
Molded Forms	Plate	- Type - Naming Category - Specification - Continuity - Tightness - Description - Split Priority - Material - Material Grade - Actual Thickness - Plate System Type - Thickness Direction - Offset - Primary Profile Orientation Forward of For Aft reference - Primary Profile Orientation Aft of For Aft reference - Primary Profile Orientation at For Aft reference - Secondary Orientation of Vertical Profiles - Secondary Orientation of Vertical Profiles on Port Starboard - Secondary Orientation of Horizontal Profiles - All user-defined attributes

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Brackets	▪ Type ▪ Naming Category ▪ Specification ▪ Continuity ▪ Tightness ▪ Split Priority ▪ Structural Priority ▪ Material ▪ Material Grade ▪ Actual Thickness ▪ Thickness direction ▪ Offset ▪ Primary Profile Orientation ▪ Primary Profile Orientation at Port Starboard Reference ▪ Secondary Orientation of Longitudinal Profiles ▪ Secondary Orientation of Vertical ▪ Profiles Forward of Fore Aft reference ▪ Secondary Orientation of Vertical ▪ Profiles Aft of Fore Aft reference ▪ Secondary Orientation of Vertical ▪ Profiles at Fore Aft reference ▪ Bracket Contour Type ▪ All user-defined attributes

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Profile	▪ Type ▪ Naming Category ▪ Continuity ▪ Continuity Priority ▪ Description ▪ Section Type ▪ Section Size ▪ Material ▪ Grade ▪ Primary Orientation ▪ Secondary Orientation ▪ Mounting Face ▪ Load Point ▪ Landing Curve Projection Method ▪ Attachment Method ▪ Start Connection Method ▪ End Connection Method ▪ All user-defined attributes

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Profile Edge Reinforcement	- Naming Category - Specification - Description - Section Type - Section Size - Material - Grade - Position - Primary Orientation - Secondary Orientation - Mounting Face - Load Point - Start Connection Method - End Connection Method - All user-defined attributes
	Seam	- Seam type - All user-defined attributes
	Profile Seam	All user-defined attributes
	Opening	- Opening Design Type - Opening Planning Method - Opening Production Method - All user-defined attributes
	Manual/Generated Logical Connections	Only user-defined attributes
	Standard Member	- Type - Type Category - Priority - Continuity Type - Continuity Priority number - Align

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Member Part	▪ Type ▪ Type Category ▪ Section Name ▪ Cardinal Point ▪ Angle ▪ Reflect ▪ End Releases ▪ Member Section Name ▪ Material ▪ Grade
	Split Connection	▪ Offset ▪ Offset Reference ▪ Distance ▪ Location Definition
	SPS Opening	▪ Cutting Limit ▪ All user-defined attributes
	Frame Connection	▪ Connection ▪ All user-defined attributes
	Built-up Member	▪ Type ▪ Type Category ▪ Priority ▪ Continuity Type ▪ Continuity Priority number ▪ Align

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Designed Member	- Cardinal Point - Angle - Reflect - Type Category - Type - Priority - Length Extension - Wide Extension - Top Flange - Depth Extension
Structural Detailing	Plate Part	- Description - Thickening Technique Requested - All user-defined attributes
	Bracket Part	- Description - Thickening Technique Requested - All user-defined attributes
	Profile Part	- Naming Category - Description - All user-defined attributes
	Edge Feature	Not Supported
	Corner Feature	Not Supported
	Sketch Feature	Not Supported
	Edge Treatment	Not Supported
	Standalone Plate Part	Not Supported

Task	Objects Supported for Modify in Copy by Family	Properties Supported
	Assembly Connections	▪ End Cut Type ▪ Split End To End Case ▪ Flip End To End Angle ▪ Chamfer Type ▪ Current Selection ▪ Question Controls ▪ All user-defined attributes
	Physical Connections	All attributes are supported
	Free End Cuts	▪ End Cut Type ▪ Chamfer Type ▪ Current Selection ▪ Question Controls ▪ Is Named By Rule
	Webcut	All attributes are supported
	Flangecut	All attributes are supported
	Smart Plate	▪ Description ▪ Thickening Technique Requested ▪ All user-defined attributes
	Slot	All attributes are supported
	Collar	All attributes are supported
Structural Manufacturing	Margin	▪ Type ▪ Value ▪ Start Value ▪ End Value
	Marking	▪ Manufacturing Marking Side ▪ Manufacturing Marking Type
	Shrinkage	▪ Shrinkage Type ▪ Primary Factor ▪ Secondary Factor

Geometry modification behavior for objects in Copy by Family

Application	Objects Supported for Modify with Geometry Modification in Copy by Family	Remarks
Molded Forms	Plate	Only boundary modifications are supported.
	Brackets	Only boundary modifications are supported.
	Profile	Geometry modification only applies to boundary and landing curve modifications.
	Seam	Geometry modification only applies to boundary and landing curve modifications.
	Seam Point	Geometry modification applies to reference plane and offset cases.
	Openings and Macro Openings	Geometry modification applies to the location change of the opening.

Solutions to common problems in Copy by Family

Problem: A problem is encountered when creating a symmetry family.

Reason: Creating a symmetry family requires an equal number of positive and negative frames for a coordinate system.

Recovery: Update the coordinate system so that there are an equal number of positive and negative frames for the coordinate system.

Problem: The following error message displays in the log file:

```
The Object is being copied before its input <NoBlk>-5WB.101-1-PS.1-1
1. Please Check the Geometry and Inputs for this Object from Source
to Target
2. Try to do Manual Copy of the Parent and Perform Find and
Correlate.
Aborted due to Issue with Input Ordering.
        !Skipped A-221LBH-1/Plate
        !Note: Above elements have been skipped, run copy a second
time
```

Reason: **Copy by Family** did not handle the ordering and wrote the message to the log file.

Recovery: Use **Copy by Family** multiple times to complete the copy.

OR

Use **Copy/Paste** to create a find to map for these objects.

Problem: Smart 3D does not find an object using **autoFind**.

Reason: The objects are not identical, or the objects are not identical crossovers.

Recovery: Perform a manual **Find** using the browser to map the objects.

Problem: Objects depending on a hull do not copy using **Across Models Family**.

Reason: Either the source or target model has more than one hull.

Recovery: Perform a manual **Find** using the browser to map the hull from the source to the target model.

Problem: The **Range Growth Value** does not impact the selection of objects.

Reason: You have modified the **Range Growth Value** after clicking **Create**.

Recovery: Refresh the family, and then select the appropriate objects.

Problem: Smart 3D does not select objects for **Modify**, even though the objects have differences in attributes and geometry.

Reason: The objects are out of the workspace, or the **Geom-Mod** option is not available on the ribbon.

Recovery: Verify that the source and target objects are in the workspace. Use the **Geom-Mod** option if a change exists in the geometry of the source object.

Problem: Objects dependent on leaf systems or parts fail to copy.

Reason: Smart 3D performs the split operation step which generates leaves on the target side during the split operation step. This step follows the Molded Forms copy step.

OR

Smart 3D performs the detailing step (which creates detailed parts on the target side) after the Molded Forms copy step.

Recovery: Perform multiple runs to copy this type of object.

Frequently Asked Questions

Q: Does Copy by Family support copy of objects created using a rotated coordinate system?

A: No, Copy by Family cannot be used to copy objects if the objects are created using rotated coordinate systems.

Q: How do I verify that the object is copied to the target?

A: Use **Audit-Tool** in **Manual Mode** to select the family and select the object to check from Smart 3D. Then, run a report to show whether Smart 3D copied the object to the target.

Q: How do I map objects that are not identical from the source to the target?

A: Use the browser in **Manual** mode to map the objects.

Q: How do I copy only certain objects in an **Across Models Family**?

A: Create a filter on the source with the objects to copy. Use that filter in the **Source Filter** box so that Smart 3D only processes the required objects.

Q: How do I copy objects that are children of an object I am not copying with **Copy by Family**?

A: Use **Copy/Paste** or **Mirror Copy** to copy the objects to the target. Then, use **Find** to map the object, and **Copy** the child objects using **Copy by Family**.

Q: How do I copy only certain objects from the source?

A: Press CTRL + Q to use the **Manual Preselect** option to manually select the objects to

copy and synchronize. Smart 3D then makes **Finish** available. Click **Finish** to copy these objects to the target. This technique also works for **Modify**.

Q: When do I use **Select All** and **Select Child**?

A: To select all the objects, click **Select All**. The selected objects display in yellow.

To clear all the objects so that you can pick the objects to copy, clear **Select All**. The objects then display in cyan. As you select the objects to copy, they display in yellow.

To select all the child objects when you select the parent, click **Select Child**.

Q: Are some objects not supported by the audit tool?

A: The audit tool does not report logical connections created by manually created features copied using **Copy by Family**.

Q: Does **Copy by Family** support all kinds of filters?

A: The command supports ship filters, SQL filters, and user filters. **Copy by Family** does not currently support asking filters**.**

Q: Which kind of coordinate systems can I use to create a symmetry family?

A: Currently, Smart 3D only supports ship coordinate systems for creating a symmetry family.

Q: What should I do if I get an HRESULT failure?

A: HRESULT failures can occur because of compilation or build issues. Try to minimize the test case by reducing the number of objects to determine the object that is causing the HRESULT failure. Then, file a TR with the simplified workflow.

Q: What should I do if I see an **Out of memory** issue during the copy process?

A: Use the **3GB** option and **GC Cache** to copy the objects from the source to the target. File a TR so that the memory issues are resolved.

Q: What is the difference between **Manual Mode** in **Find** and **Manual Find** in the browser?

A: Smart 3D uses both options to map source to target objects. In **Manual Mode**, you can resolve one object at a time. You can use **Manual Find** to resolve multiple objects at a time.

Q: What is the custom filter option in **Copy by Family**?

A: You can select either a workspace filter or any specific filter for performing the following actions:

- **Find/Create/Modify/Delete** option in **Copy by Family**.
- Select objects for processing that are common to the workspace by using the **Custom Filter** option.

Q: What issues do I face when I use coordinate systems with different frame spacing in **Across Models Family**?

A: **Issue 1** - If you create objects using one coordinate system but create the family using another coordinate system, Smart 3D creates substitute planes at an improper location, because the frame spacing is different for both of the coordinate systems. Because of this, use the same coordinate system for both the family and the objects.

Issue 2 - If you create the hull and range family with coordinate systems having different frame spacing, hull mapping fails. This also causes all of the dependents of the hull to fail to copy.

Q: Does **Copy by Family** support modification (**Property** or **Geometry**)?

> **A**: No, you cannot propagate feature modification to the target by using the **Copy by Family** modify operation. You can achieve this by updating the feature count on the source by adding or removing features on the source. **Copy by Family** picks up this feature count as a difference and deletes and recopies all of the features to the target.

Q: What kind of advanced plate system (APS) plates are supported using **Copy by Family**?

> **A**: **Copy by Family** supports the following advanced plate systems:

- Plate systems created using APS polyline plate macros
- Plate system created using APS corrugated plate macros
- Plate system created using APS swage plate macros

Q. What is the **System Hierarchy** option in **Copy by Family**?

> **A. Copy by Family** provides target parent system options on the **Select Copy Family** dialog box so that the target objects are copied under the given parent system. This makes target objects easier to locate.

Q: What is **Copy/Paste**? How can I use this command in conjunction with **Copy by Family**?

> **A**: **Copy/Paste** lets you copy and paste objects from the source to the target by taking inputs from the **Paste** dialog box.

> If an object fails to copy from the source to the target using **Copy by Family**, then all the dependent objects also fail to copy because the parent is missing on the target. Use **Copy/Paste** to copy such objects, and perform an auto or manual **Find** to map the source to the target object. Then, use **Copy by Family** to copy the dependent objects.

Q: What should I do to decide if a problem is related to **Copy by Family**?

> **A**: Perform the following actions:

a. Verify that the object is supported by **Copy by Family**. For more information, see *Objects and behaviors supported for Copy by Family* (on page 347).

b. If the object is supported, check for a **ToDo** list entry.

c. Try to **Edit** the object.

d. Try to **Copy/Paste** the object.

e. Try **Model Data Reuse** for the object.

> If all the previous actions are successful, then the issue is related to **Copy by Family**.

Q: What is the port mapping issue in **Copy by Family**?

> **A**: If the following message displays in the finish log, then this is a port mapping issue in **Copy by Family**.

```
WARNING :  Cannot resolve input mapping for object->
IJPortSrc-SF_!BLFX_0_0_8_<>_{}_()||D3-DCK
!Skipped F 70.000 m-1TBH-1/Plate
!Please review with support
```

Possible reasons:

- Smart 3D did not copy an object that the port needs.
- An ordering issue from **Copy by Family** exists.

- A port mapping issue from **Copy by Family** exists.

  Use **Copy/Paste** to copy the object, and then use **Auto** or **Manual Find** to map the objects using **Copy by Family**.

Q: What are the implications of using **Commit** in rules?

A: Avoid using **Commit** and client-tier references in rules. Using these references can cause Smart 3D to exit.

S ECTION 26

Copy By Family Synchronize

The **Edit** > **Copy by Family Synchronize** command opens the **Copy by Family** ribbon with the last used family set as the default family.

NOTE This command can be used if you want to perform **Copy by Family** for families that already have been created. If no families exist, a message is displayed informing you that there are "**No Copy Families Defined**".

See Also

Copy By Family Log Files (on page 342)
Select Copy Family Dialog Box (on page 337)
Similar Family Properties Dialog Box (on page 330)
Symmetry Family Properties Dialog Box (on page 333)

Copy By Family Audit Report

The **Tools > Copy by Family Audit Report** command allows you verify that the source has been copied to the target by highlighting selected source and targets after a copy has been performed.

Copy By Family Audit Report Ribbon

Edit file report properties

Opens the **Settings** dialog box in which you set the path to the location where you want to save the generated report. By default, the path for all the reports is set to C:\Documents and Settings*[User name]*\Smartmarine3d\Report Output.

NOTE Currently, three kinds of reports are generated. These reports are generated in the **Microsoft Excel** workbook format and can be saved for future reference.

- **Object Summary Report** - Provides information on object details.
- **Property Summary Report** - Verifies that the properties of source and target objects match.
- **Property Compare Report** - Provides a detailed description of the Property Summary Report.

Select Objects

Select objects manually from **Workspace Explorer** or from the graphic view when **Select Mode** list is set to **Manual**.

Run Report

Allows you process all the objects and displays the report. The **Run Report** option is not available if it is **Manual** unless an object is selected manually.

Finish

Closes the reports. This option is only available after you run a report.

Reject

Rejects all selected objects.

Accept

Accepts all selected objects.

Report

Select **Object Summary Report** or **Property Summary Report.** The **Object Summary Report** is available by default.

Select Mode

Lists the following modes:

- **Manual** - Selects objects manually from the **Workspace Explorer** or graphic view.

- **Auto Select Family Objects** - Selects all objects associated with the selected family for processing.

- **Auto Select Family Objects in Workspace** - Selects all objects associated with the selected family which are in current workspace.

- **Open Existing Reports** - Displays existing reports.

- **Manual (recursive)** - This is same as **Manual** mode except that in this case, when an object is selected, its children also get selected.

Copy Family

Select the family on which you want to run the report. Acceptable values are **Copy Across Models Family**, **Similar Family**, or **Symmetry Family**.

Object Summary Report

The **Object Summary Report** has three worksheets.

- Worksheets with object information. These worksheets are named in the following format - "Source group-Target group".

- **Configuration** worksheet

- **Defective Entities** worksheet

 NOTE The report has multiple worksheets for multiple targets.

Worksheet with Object Information

In the worksheet with object information, you can find the following information:

Status

Displays the status of the object. If the object is copied, the status is **OK.** If the object is not copied, the status is **Check**.

 NOTE When the status is **Check**, you need to check for any discrepancies in the source and the target. The entire row is highlighted in yellow. The parameters which fail to match are highlighted in red.

Object Type

Displays information on the type of object.

Source Object Name

Displays the name of the source object.

TDL

Indicates whether this object (**Source** or **Target**) is present in the To Do List.

LC's

Displays the count of logical connections.

AC's

Displays the count of assembly connections.

PC's

Displays the count of physical connections.

Fetr's

Displays the number of features available for the object.

Detailed

Displays a detailed status on the object.

Copied

Specifies whether the object has been copied successfully.

Target Object Name

Displays the name of the target object.

Configuration Worksheet

The **Configuration** sheet provides you with the following information:

Generation Date

Displays the date when the report was generated.

UserID

Displays user identification information.

Selected Mode

Specifies the mode selected.

Compute Time

Displays the time taken (in seconds) for generating the report.

Number of Objects

Displays the number of objects selected.

Defective Entities Worksheet

If there is any information about defective entities, it displays on the **Defective Entities** sheet.

NOTE When a cell in any row is selected in the report, the corresponding source and target objects are highlighted in the graphic view.

You can also run the **Object Summary Report** on a **Copy Across Model** family.

Property Summary Report

The **Property Summary Report** has two work sheets - **Parameter Summary** and **Defective Entities**.

In the **Parameter Summary** sheet, you can find the following information:

Status

Displays the status on the properties of the source and target objects. If all the properties for the source and target match, the status is **Ok**. If any of the properties do not match, the status is **Check**.

NOTE If the status is **Check**, the entire row is highlighted in yellow. For multiple targets, the status is **Check** if the properties of the source object fail to match with any of the targets.

In this case, the cell corresponding to that target is highlighted in red.

Object Type

Displays information on the type of object selected.

Source

Displays the name of the source object.

Show Details

Describes the **Property Summary Report** in detail**.**

Target

Displays the name of the target object.

NOTE In all the reports, any discrepancies in source and target are highlighted in red.

Property Compare Report

The **Property** sheet lists all of the properties of the source and targets and their status. In the **Property** sheet, you can find the following information:

Status

Displays the status for each of the properties of the source and target objects.

NOTES

- If the status is **Check**, it indicates that the property of the source and target has failed to match. The entire row is highlighted in yellow. If the property matches, the status is **Ok**.

- When the status is **Check**, a comment is displayed on the target box (when you move the pointer over the target box) indicating the expected value that is needed for the properties to match.

- When the target value is expected to be different (in cases, when target attribute value is defined by a rule), the status is **Ok**.

Property

Displays information on any property associated with the selected objects.

Source

Displays the name of the source object.

Target

Displays the name of the target object.

NOTE If the objects being processed for a particular report do not belong to the selected family, then the following message displays: **There are some objects not associated with the family. Hence these objects will not be reported in the Report.**

What do you want to do?

- *Generate an object summary report in manual mode* (on page)

- *Generate an object summary report in auto select family objects mode* (on page)

- *Generate an object summary report in auto select family objects in workspace mode* (on page 375)
- *Generate a property summary report in manual mode* (on page 375)
- *Generate a property summary report in auto select family objects mode* (on page 375)
- *Generate a property summary report in auto select family objects in workspace mode* (on page 376)

Generate an object summary report in manual mode

You can generate the object summary report in manual mode to generate the report for required objects in family defined in the workspace.

1. Click **Tools > Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*

2. In the **Report** box, select **Object Summary Report**.

3. In the **Select Mode** box, select **Manual**.

4. In the **Copy Family** box, select the family on which to run the report.

5. Select the objects from **Workspace Explorer** or the graphic view.

6. Click **Run Report** to display the **Object Summary Report**.

Generate an object summary report in auto select family objects mode

If you want to select all the objects related to the family without regard to the defined workspace (for example, all the related objects in the database), you can generate the report in auto select family objects mode.

1. Click **Tools > Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*

2. In the **Report** box, select **Object Summary Report.**

3. In the **Select Mode** box, select **Auto Select Family Objects**.

4. In the **Copy Family** box, select the family on which to run the report.

5. Click **Run Report** to display the **Object Summary Report**.

Generate an object summary report in auto select family objects in workspace mode

You can generate the object summary report in auto select family objects mode to generate the report for required objects in a family defined in the workspace.

1. Click **Tools** > **Copy By Family Audit Report**.

    the Audit Report ribbon displays.

2. In the **Report** box, select **Object Summary Report**.

3. In the **Select Mode** box, select **Family Objects in Workspace**.

4. In the **Copy Family** box, select the family on which to run the report.

5. Click **Run Report** to display the **Object Summary Report**.

Generate a property summary report in manual mode

You can generate the property summary report in manual mode to generate the report for specific objects in the family defined in the workspace.

1. Click **Tools > Copy By Family Audit Report**.

    The Audit Report ribbon displays.

2. In the **Report** box, select **Property Summary Report.**

3. In the **Select Mode** box, select **Manual**.

4. In the **Copy Family** box, select the family on which to run the report.

5. Select the objects from **Workspace Explorer** or graphic view.

6. Click **Run Report** to display the **Property Summary Report**.

Generate a property summary report in auto select family objects mode

If you want to select all the objects related to the family without regard to the defined workspace (for example, all the related objects in the database), you can generate the report in **Auto Select Family Objects** mode.

1. Click **Tools > Copy By Family Audit Report**.

    The Audit Report ribbon displays.

2. In the **Report** box, select **Property Summary Report.**

3. In the **Select Mode** box, select **Auto Select Family Objects**.

4. In the **Copy Family** box, select the family on which to run the report.

5. Click **Run Report**.

    The Property Summary Report displays.

Generate a property summary report in auto select family objects in workspace mode

You can generate the property summary report in auto select family objects mode to generate the report for required objects in a family defined in the workspace.

1. Click **Tools** > **Copy By Family Audit Report**.

   *The **Audit Report** ribbon displays.*

2. In the **Report** box, select **Property Summary Report**.

3. In the **Select Mode** box, select **Family Objects in Workspace**.

4. In the **Copy Family** box, select the family on which to run the report.

5. Click **Run Report**.

   *The **Property Summary Report** displays.*

SECTION 28

Place Manual Chamfer

Creates a manual chamfer between two detailed parts. This command supports creation of a chamfer between the following part types:

- Detailed plate parts
- Standard member parts
- Detailed edge reinforcements
- Any combination of the previous parts

Place Manual Chamfer Ribbon

Controls options for placing a chamfer between two detailed part ports. The parts do not have to be connected.

Properties

Displays the **Chamfer Properties** dialog box, which you use to view and modify the properties of the chamfer before it is written to the database. For more information, see *Chamfer Properties Dialog Box* (on page 432).

Chamfer Method
Displays images corresponding to supported methods for creating a manual chamfer.

- **Bounded Plate Chamfer** - Initiates chamfer creation by selecting part, face, and face.

- **Unbounded Plate Chamfer** - Initiates chamfer creation by selecting part and face.

- **Edge Reinforcement Chamfer** - Initiates chamfer selection by selecting an edge reinforcement part and face.

Select Parent Part

Select the part that contains the first port to use in creating the chamfer. This part is the parent of the chamfer.

Select Port to be connected from Parent Part

Specifies a port from the parent part. This port is used as the first input to the chamfer for bounded or unbounded chamfers.

Select Other Port to connect to

Specifies a port from another part that overlaps the port from the parent part. This port is not required for an unbounded chamfer.

Select Port on End Cut

Specifies a port on the Edge Reinforcement end cut of the chamfer.

Finish

Creates the chamfer.

Chamfer Class

Specifies the chamfer root class for the chamfer.

Chamfer Item

Specifies the chamfer.

Rule Based

Enables or disables the **Chamfer Item** list. Selecting this check box disables the **Chamfer Item** list.

Extension Type

Allows you to select the length options for a chamfer. Select **Overlap**, **Full Edge**, or **Extend by Value**.

- **Extension 1** - Specifies the start of the extension length. This option is available only when **Extend by Value** is selected.

- **Extension 2** - Specifies the end of the extension length. This option is available only when **Extend by Value** is selected.

What do you want to do?

- *Place a manual chamfer* (on page)
- *Modify a chamfer* (on page 379)
- *Delete a smart occurrence object* (on page 58)
- *Modify a smart occurrence object* (on page 58)

Place a manual chamfer

1. Click **Place Manual Chamfer** .

2. Select a chamfer creation method from the **Chamfer Method** list.

    - **Bounded Plate Chamfer**

    - **UnBounded Plate Chamfer**

    - **Edge Reinforcement Chamfer**

3. Select the part that contains the first port.

4. Select a port from the selected part.

5. For a non-single port chamfer, select the overlapping port from another part.

    TIP Only ports that have overlapping geometry and do not currently have a chamfer are available for selection.

    The software creates a chamfer between the selected ports.

6. Select an extension type for the chamfer. If **Extend by Value** is selected, type the start and end extension values into **Extension 1** and **Extension 2**.

7. Select the root class from the **Chamfer Class** list.

8. Click **Finish**.

Modify a chamfer

1. On the vertical toolbar, click **Select** .

2. In **Locate Filter**, select **Features**.

3. In the graphic view or in the **Workspace Explorer**, select the chamfer to modify.

4. Modify the chamfer using the **Place Manual Chamfer** ribbon.

S E C T I O N   2 9

Board Management Service

The **Tools** > **Board Management Service** command determines structural part and seam symmetry about the center plane.

- Naming rules can consider symmetry information. For example, you can include a suffix indicating the symmetry property value for parts. Naming rules can also sort parts within a block by the symmetry value before assigning a part name index.

- The board property can be used in the nesting and cutting processes by possibly nesting and then cutting symmetrical parts at the same time, or nesting the symmetrical parts on the same raw material stock if the parts are in the same block. The Board Management Service determines if parts are symmetrically similar. Structural Manufacturing determines if the parts are symmetrically identical for manufacturing purposes.

- Unfolding algorithms use seam symmetry information when they calculate shapes for plates that cross the centerline.

- Some assembly planning tasks use the board property to automate creating and maintenance of assembly hierarchies for symmetrical parts.

- Reports include part symmetry information.

- Seam board information supports shell expansion drawing requirements. The shell expansion drawing for the entire ship displays and labels both port and starboard shell seams in one view.

- You can choose to manually override board management review for an object. If you choose this option, then board management disregards the object for subsequent review.

Whether you choose to run the Board Management service periodically or continuously depends upon your processes and workflow. If you need to see the symmetry immediately, then running this process continuously on a remote server may best suite your needs. If you do not need immediate results, then you can run it periodically on your local computer at times that best fit your workflow.

NOTE This property affects how Smart 3D names parts. It also affects how the parts are handled in Planning and Structural Manufacturing. Because of this, we recommend that you run the Board Management service before you generate drawings, bills of materials, or manufacturing output.

Board Management Service Console

Specifies the settings for running the board management service.

Server

Displays the name of the server on which the board management service runs.

Started by

Displays the user name of the person who started the board management service.

Seams Selected for Evaluation

Hull Seams

Indicates that seams on hull surfaces should be evaluated for symmetry.

Interior Seams

Indicates that seams on interior surfaces should be evaluated for symmetry.

Filters Applied to Target Selection

by Type

Indicates that parts should be filtered by type. The **Type** property is set on the **System Properties** dialog box.

by Naming Category

Indicates that parts should be filtered by naming category. The **Naming Category** property is set on the **System Properties** or **Part Properties** dialog box.

by Region

Indicates that parts should be filtered by region or bounding box. The bounding box is defined by the minimum and maximum x, y, and z locations of the object.

Growth

Specifies the amount to expand or shrink the region before filtering. The intent is to find parts that are approximately the same size and in approximately the same (mirrored) location.

Previous Part List Date/Time

Displays the date and time of the previous generation of the list of new and modified parts.

Reset Date/Time to

Replaces the start value of the last time the service was run. This time is used to search for new and modified parts.

Processing Status

Displays information about the number of parts processed and an estimation of the time remaining.

Start

Starts the board management service.

Stop

Quits the board management service without completing the process.

Close

Saves the values and closes the dialog box.

Run the board management service

1. Click **Tools > Board Management Service**.

    TIP Specify the seam types to include.

2. Specify the filters to include.

3. If necessary, reset the start date and time.

4. Click **Start**.

Submit Batch Job

Tools > **Submit Batch Job** runs batch processes using the Intergraph Batch Services framework.

This command runs the following types of batch jobs:

Detailing:

- Detail parts
- Undetail parts
- Update reports

Production:

- Generate manufacturing parts
- Generate XML output files
- Manufacturing Service Manager update

Custom batch processes:

- BO recompute
- Other custom batch processes that you define.

What do you want to do?

Run a detailing batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Detailing** from the **Batch Process Type** list.

   *Smart 3D displays the available detailing batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide.*

Run a production batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Production** from the **Batch Process Type** list.

   *Smart 3D displays the available production batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide.*

Run a custom batch process

1. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

2. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

3. Select **Custom Batch PRocesses** from the **Batch Process Type** list.

   *Smart 3D displays the available custom batch processes in the **Select Batch Process** list.*

4. Select the process to run from the **Select Batch Process** list, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

5. Specify the **Batch Input Type**, and add inputs as necessary.

6. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

7. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide.*

Add a custom batch process

1. Add the ProgID and job description of the custom batch process to the <Product Folder>\CommonShip\SOM\Client\Xml\CustomBatchJobDetailsConfiguration.xml file.

   For example:

   ```
   <JOB>

       <JobType>CustomBatchProcessD</JobType>

       <JobType_LocaleID>103</JobType_LocaleID>

       <JobDescription>Description of
   CustomBatchProcessD</JobDescription>

       <JobDescription_LocaleID>104</JobDescription_LocaleID>

       <JobProgId> ProgID of CustomBatchProcess </JobProgId>

       <IsModelSpecific>False</IsModelSpecific>

   </JOB>
   ```

2. Click **Tools > Submit Batch Job**.

   *The **Generic Batch Command** dialog box displays.*

3. Select the batch process to run from the **Batch Process** list. If the process you want to run is not in the list, select **More** from the **Batch Process** list.

   *The **Batch Process Selection** dialog box displays.*

4. Select **Custom Batch Process** from the **Batch Process Type** list.

   *The available custom batch processes display in the **Select Batch Process** list.*

5. Select the custom process to run, and click **OK**.

   *Smart 3D returns to the **Generic Batch Command** dialog box.*

6. Specify the **Batch Input Type**, and add inputs as necessary.

7. Click **Schedule**.

   *The **Schedule [Task] Batch Job** dialog box displays.*

8. Set up the parameters to run the batch job. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide.*

Generic Batch Command Dialog Box

Displays the controls used to submit a generic batch command.

Batch Process

Specifies the batch process to run. The list contains recently-selected job types. Click **More** to display the **Batch Process Selection** dialog box, and select a process that is not on the list. For more information, see *Batch Process Selection Dialog Box* (on page 386).

Batch Input Type

Indicates the input type for the batch process.

- **Filter** indicates that you are providing a filter as an input to the batch job.

- **Assembly** indicates that you are providing an assembly as input to the batch job.

- **Object** indicates that you are providing a set of objects as input to the batch job.

 Add

Adds inputs to your batch job.

- If **Filter** is selected, this button displays the **Select Filter** dialog box so that you can add filters as inputs to your batch job.

- If **Assembly** is selected, this button displays the **Select Assembly** dialog box so that you can add assemblies as inputs to your batch job.

- If **Object** is selected, this button is not available. To add objects as inputs, select them from the model or the **Workspace Explorer**.

 Remove

Removes the selected inputs from the list. The software only removes the inputs that have the associated check box selected.

Share

Saves the inputs for the next batch process. If this check box is cleared, Smart 3D clears the inputs when you select a different batch process.

Schedule as Multiple Jobs

Processes each item in the inputs list as a separate batch process. If this check box is cleared, Smart 3D processes all of the inputs as a single batch process.

Schedule

Displays the **Schedule [Task]** dialog box so that you can schedule the batch process. For more information, see Schedule [Task] Dialog Box in the *Common User's Guide*.

Cancel

Exits the command.

Batch Process Selection Dialog Box

Displays the controls used to specify the type of batch process to run.

Batch Process Type

Displays the general batch process types. Select a type to display the available batch process of that type in the **Select Batch Process** list.

Select Batch Process

Displays the available batch process associated with the selected type.

Delay Settings (Tools Menu)

Tools > Delay Settings specifies whether changes made in Molded Forms are immediately updated in Structural Detailing or Structural Manufacturing. When the delay is active, changes to the Molded Form systems appear in Molded Forms, but changes needed to the corresponding Structural Detailing and Structural Manufacturing objects are put on the **To Do List.** When the delay is not active, the changes needed to the corresponding Structural Detailing and Structural Manufacturing objects are propagated immediately.

☐ NOTE This setting is saved in, and applies only to, your session file. Therefore, it is possible that objects that you expect to appear in the **To Do List** propagate instead because they are used in someone else's session where the delay setting is off.

This command opens the **Delay Operations** dialog box.

What do you want to do?

- *Turn on delay operations* (on page)
- *Update delayed objects* (on page 387)
- *Turn off delay operations* (on page 387)

Turn on delay operations

1. Click **Tools > Delay Settings**.
2. Click the **Delay Settings** tab.
3. Turn on the **Delay Structure Operations** options that meet your requirements.
4. Click **OK**.

Update delayed objects

1. Click **Tools** > **Delay Settings**.
2. Click the **Update To Do Records** tab.
3. Define the record filter as required.
4. Type a file name and folder path for the error file.
5. Click **Update Objects**.

Turn off delay operations

1. Click **Tools** > **Delay Settings**.
2. On the **Delay Settings** tab, click **No Delay**.
3. Click **OK**.

Delay Operations Dialog Box

Displays settings for the **Delay Settings** command.

Delay Settings Tab (Delay Operations Dialog Box)

Specifies the delay operation, if any, that you want taken.

Report Data Generation

Disable Report Data

Select to disable the report data generation.

Delay Structure Operations

No Delay

Select this option to have changes to the Structural Manufacturing and Structural Detailing objects propagated when the Molded Form object changes.

Delay Molded Forms

Select this option to have changes to the Molded objects sent to the **To Do List** instead of being propagated.

Delay Struct Detailing

Select this option to have changes to the Structural Detailing objects sent to the **To Do List** when the Molded Form object changes.

Update To Do Records Tab (Delay Operations Dialog Box)

Controls how objects that you have delayed to the **To Do List** are updated.

To Do Record Filters

User Name

Select the user name to filter the **To Do List** records by.

Dates

Define the starting (**From**) and ending (**To**) dates to filter the **To Do List** records by.

Order

Select the order in which to process the **To Do List** records. Select **None**, **Optimal**, **Ascending**, or **Descending**.

TIP Select **Optimal** to allow the software to pick the order of updating that most logically clears records from the **To Do List**.

Type

Select which type of **To Do List** records to process. Select **Errors**, **Warnings**, **Out-of-Date**,

---

Out-of-Date Reporting, or **Out-of-Date Molded Forms**.

Structure to Update

Select **All**, **Hull**, or **Select a Filter.**

- **All** or **Hull** updates the objects accordingly.

- **Select a Filter** launches additional options, **CreateNewFilter** and **More(Select from Existing Filters)**. Both options require you to select a filter before the system can update the objects.

Update To Do Records

Update Objects

Click to update all **To Do List** records that satisfy the filter you have defined. Objects are updated in the selected order.

Cancel Update

Stops the update process.

Batch Update

Click to update all **To Do List** records that meet the requirements for the filters that you have defined through the **Batch** process. The system updates objects in the optimal order.

Log

Log File

Type a file name and folder path for the error log.

Enable Logging

Turn on to generate the log file.

View Log File

Click to display the error log.

To Do Delay Record Detail Tab (Delay Operations Dialog Box)
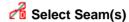
Displays detailed information about the delayed **To Do List** records.

S ECTION 32

Split Seam

 Splits a seam by intersecting seams. After you select the seam to be split, the software displays a list of all intersecting seams. You can then select which splits you want performed.

Split Seam Ribbon

Displays the options that you use to split a seam.

 Select Seam(s)

Specifies the seam to be split.

 Reject

Rejects the seam selection and clears the **Split List**.

 Accept

Opens the **Split List** dialog box, which shows all seams intersecting the selected seam. If the dialog box is already open, refreshes the **Split List** to update the list of intersecting seams for the selected seam. For more information about the **Split List** dialog box, see *Split List Dialog Box (Split Seam)* (on page 391).

Plate Split Delay

When selected, delays the split plate update. The new position and boundaries of the design seam appear in the graphic view when this option is selected. However, the plate being split is placed on the To Do List with the message "Plate System split was stopped due to delay split" with a status of "Out of Date." You must go to the To Do List and update the plate system at a later time. This option is available only when you modify a design seam. If you do not select this option, the plate is updated as soon as you move the design seam, which can take time on a complex plate system, but no To Do List item is created for the plate.

Split a seam by intersecting seams

1. Click **Split Seam** on the vertical tool bar.

2. Click **Select Seam(s)** .

3. Select the seam that you want to split. The seam must be intersected by one or more seams.

4. Click **Accept** .

   *The **Split List** dialog box is displayed, showing all intersecting seams for the selected seam.*
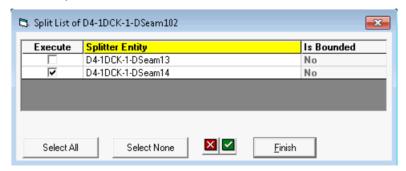
5. In the **Execute** column, select the intersecting seams to split the seam, and click **Finish**.

   *The software immediately processes the splits. When it is completed, the **Split List** dialog box remains open and the split list is cleared.*

6. To split another seam, click **Select Seam(s)** in the ribbon to select the seam to split, and click **Accept** (either in the ribbon or in the **Split List** dialog box) to refresh the **Split List**. Select the splits to perform and click **Finish**.

7. Click **Reject** (either in the ribbon or in the **Split List** dialog box) to clear the **Split List** and clear the selection of the seam.

8. Close the **Split List** dialog box when you finish.

Split List Dialog Box (Split Seam)

This dialog box opens during the **Split Seam** command and specifies the intersecting seams used to split the selected seam.

Execute

Select which splits to process.

Splitter Entity

Displays the seams intersecting the selected seam.

Is Bounded

Specifies if the intersecting seam is bounded to the selected seam.

Select All

Selects all the splitter entities.

Select None

Clears all the splitter entities.

 Reject

Rejects the seam selection and clears the **Split List**.

 You can directly click **Select Seam(s)** in the ribbon to select another seam to split without closing the **Split List** dialog box.

 Accept

Refreshes the **Split List** to update the list of intersecting seams for the selected seam.

Finish

Splits the seam using the parameters you have defined. When the split process is complete, the **Split List** is cleared.

Check Manufacturability

Tools > **Check Manufacturability** (**Tools > Check Hole** in the **Hole Management** task) analyzes objects in the current workspace and reports the objects that will be difficult or impossible to manufacture according to standards defined in the reference data.

Check Manufacturability Ribbon

Provides the following options to check that objects in the model can be manufactured and to generate production information.

Manufacturability Checking Settings

Shows the rules for the checking process. You can select any or all of the rules. For more information, see *Manufacturability Checking Settings Dialog Box* (on page 394).

Check Manufacturability

Starts the checking process. If inconsistencies are found, the message **Manufacturability errors/warnings have been encountered** displays.

Show Manufacturability Inconsistencies

Displays the objects with manufacturing issues. The list includes the issue severity, the assembly name, a description of the assembly, and the name of the rule which detected the manufacturing issue. For more information, see *Show Manufacturability Inconsistencies Dialog Box* (on page 395).

Close

Exits the command.

Submit Job

Displays a message box asking if you want to execute the repair rules subsequently. If you click **Yes**, the **Schedule Planning Check Manufacturability** dialog box displays so that you can schedule the execution of the repair rules at some other time. For more information, see *Schedule [Task] Dialog Box (on page 340)* in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

NOTES

- Learn more about SmartPlant Batch Services in the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

- The **Submit Job** button is enabled only when Batch Services is installed and started on your computer.

- The data on objects that cannot be manufactured is stored in the session file, not in the model. Problem objects that apply to the Workspace are retrieved when you select the **Check Manufacturability** command.

What do you want to do?

- *Check objects for manufacturability* (on page 393)
- *Schedule check manufacturability using Batch Services* (on page 393)
- View the *Manufacturability Checking Settings Dialog Box* (on page 394)
- View the *Show Manufacturability Inconsistencies Dialog Box* (on page 395)

Check objects for manufacturability

1. Select objects for checking in the model or in the **Workspace Explorer**.
2. Click **Tools > Check Manufacturability** (**Tools > Check Holes** in Hole Management).
3. On the ribbon, click **Manufacturability Checking Settings** 🖳.
4. Specify the rules for the checking process.
5. On the ribbon, click **Check Manufacturability** 🔃 to start the process.
6. On the ribbon, click **Show Manufacturability Inconsistencies** 🗂.

   A list view displays the objects that were found to have manufacturing problems.
7. Select one row of the list at a time to view the inconsistency and solution for an object.
8. Repair the object manually, or click **Repair** if an **Action Tool** is available.

Schedule check manufacturability using Batch Services

1. Select objects for checking in the model or in the **Workspace Explorer**.
2. Click **Tools > Check Manufacturability**.
3. On the ribbon, click **Manufacturability Checking Settings** 🖳.
4. Specify the rules for the checking process.
5. Click **Submit Job**.
6. Click **Yes** to schedule the job for another time.
7. Complete the **Schedule Check Manufacturability** dialog box to configure batch processing, and then click **OK**.

   The software displays the message, "This job has been scheduled to run on the chosen server."
8. Click **OK** to dismiss the message.

📝 **NOTES**

- For more information about SmartPlant Batch Services, see the *Batch Services User's Guide* and the *Batch Services Quick Start Guide*.

- After the job is successfully submitted, you can see the PlanningCheckMfcty_BatchJob.xml file in the system temp folder (%temp%). This file contains the required information to run the batch job.

- If any inconsistencies are observed during the execution of the check manufacturability rules that you selected, information about the name of the object, manufacturability rule ProgID, and repair rule ProgID are written to the PlanningCheckMfcty_InConsistencies.log file in the system temp folder (%temp%).

Manufacturability Checking Settings Dialog Box

Sets options for the **Check Manufacturability** process.

Task List

Displays the list of tasks that contains the **Check Manufacturability** rules defined in the catalog. These tasks allow you to filter the check rules based on the selected task in the **Task List**.

Rule

Select which rules to process check manufacturability against. There are different rules based on the task selected in the **Task List** option. Contact Intergraph Support http://support.intergraph.com for help with customizing the XML data file.

- The **Hole Management** manufacturability checking process includes checking the distance between hole traces; the distance between the hole trace and plate edges (outside), brackets and coamings; and the distance between the hole trace and seamlines, profiles, and openings. Because hole traces can affect the stress of the deck or bulkhead on which they are placed, it is important to check holes before cutting them. For more information about hole management checks, see Check Hole Rules in the *Hole Management Reference Data Guide*.

- The **Piping** manufacturability checking process includes software simulators for cutting, bending, and coating pipe. An XML data file controls the piping simulators. You can customize this file to reflect the requirements, standards, and equipment of your facility. For more information about piping checks, see Piping Check Manufacturability in the *Piping Reference Data Guide*.

- The **Planning** manufacturability checking process includes software simulators for planning joints, weld bevels, and slot types on structure. You can customize the planning simulators rules to reflect the requirements, standards, and equipment of your facility. For more information about planning checks, see Check Manufacturability Rules in *Planning Reference Data*.

- The **Structural Detailing** checking process includes tee weld chamfer checks based on changes to plate thickness, slot opening angles, and end cut type. For more information, see Check Manufacturability Rules in *Structural Detailing Reference Data*.

- The **Structural Manufacturing** checking process includes margin check on the same port, manufacturing plat and profile check, and shrinkage check.

ProgID

Programming ID of the rule that found the inconsistency. Matches the **Rule ProgID** of the rule on the *Show Manufacturability Inconsistencies Dialog Box* (on page 395) .

Select All

Specifies all available rules.

Clear All

Clears all available rules. You can select rules individually in the list view by clicking the boxes beside the rule names.

Stop checking if a manufacturability error is found
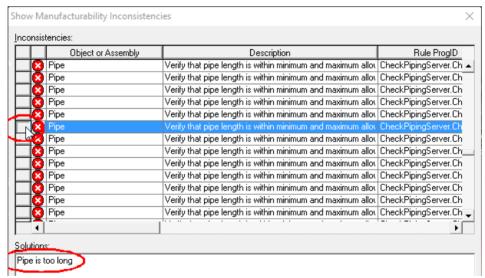
Halts rule processing upon error detection.

Show Manufacturability Inconsistencies Dialog Box

Displays and repairs inconsistencies found by the rules of the **Check Manufacturability** process.

Inconsistencies

A list view of all inconsistencies. Click the button in the far left column to see the possible solution. Solution descriptions are listed below.

⚠ **Warning**

A minor manufacturability problem has been found. The problem does not have to be repaired.

❌ **Error**

A major manufacturability error has been found. The error must be repaired.

Object or Assembly

Name of the object with an inconsistency.

Description

Description from the rule of the inconsistency

Rule ProgID

Programming ID of the rule that found the inconsistency. Matches the **ProgID** of the rule on the **Manufacturability Checking Settings Dialog Box** (on page 394) ⬜.

Action

The type of action that is available in the software to repair the inconsistency. Blank if no software action is available and the repair is done manually.

- **COM Repair Object** - Software code that can be run.
- **SQL Script** - SQL Script that can be run.

Action Tool

ProgID of the COM Repair Object or name of the SQL Script. Blank if **Action** is blank.

Solutions

Describes the details of an inconsistency and the repair that you need to perform for one selected object from the list.

Repair

Runs the **Action Tool** to perform a repair, if one is available for the rule.

Fit

Fits one or more selected objects from the list view in the active graphic view.

Clear

Clears all inconsistencies and closes the dialog box. You must click **Check Manufacturability** 🖼 again to display the remaining inconsistencies.

Piping Inconsistencies Solution Messages

Pipe and Spool Length Messages

Pipe is too long

Pipe exceeds standard ordering length for pipe of this size, as defined in the pipe catalog. For bent pipes the length check includes any extra pipe that had to be added at the ends or between bends for clamping (see "Extra pipe must be added at end…" and "Insufficient straight pipe between bends…" messages below.). If the pipe is bent, the elongation of the pipe during bending will be taken into account when reporting this error.

Pipe is too short

Pipe length is less than the company standard for pipes welded on both ends.

Spool is too long

The overall length of the spool exceeds company standards. The length is measured along the main axis of the spool.

Spool is too wide

The "width" in the spool is measured perpendicular to the main axis. This message indicates that the width exceeds company standards. Checks for surface treatment tank size are done separately.

<u>**Pipe Bending Messages**</u>

Bend angle too large

A pipe bend exceeds the maximum bend angle that the bending machine can make.

Bend angle too small

A pipe bend angle is less than the company standard for bends. That is, the pipe is almost straight.

Bends have different radii, not allowed

Pipe has two or more bends, and they do not all have the same bend radius specified.

Bend radius too large or points too close together

The pipe geometry is physically impossible. This is an extreme case of the "insufficient straight pipe between bends…" problem. This problem may be fixed by moving a bend or using a smaller bend radius.

Extra pipe must be added at end, pipe end too short for bending machine to clamp onto

The bending machine needs a sufficiently long straight section at the start and end of the pipe to clamp onto (start), and support (end). This is a warning; bending can be done with a longer piece of pipe, with the extra pipe cut off afterwards.

Insufficient straight pipe between bends for bending machine to clamp onto

The bending machine needs a sufficiently long, straight section between bends to clamp the pipe. This problem may be fixed by moving a bend, by using a smaller bend radius, or by splitting the pipe and putting a joint between the bends.

No pipe bending machine for pipe diameter and bend radius

None of the pipe bending machines listed in the pipe shop XML data file have bend dies listed for the pipe diameter and bend radius of this pipe.

Pipe hits machine or floor during bending

The pipe cannot be bent because the free end or part of the pipe would hit the bending machine or the shop floor during bending. The simulator will have evaluated bending the pipe starting from either end before reporting this error, and will have checked all available bending machines with bend dies of the right size.

NOTE If the pipe has flanges on either end, and the **Flange Welding** check option has been selected, the simulator also checks for attached flange (if any) hitting the machine during bending.

<u>**Pipe and Spool Complexity**</u>

Too many bends in pipe

The number of bends in the pipe exceeds the company standard. This limit is normally set to maintain dimensional accuracy standards because of accumulated error during bending.

Too many branches

The number of branches off of a main exceeds the company standard limit.

Too many planes in spool

The spool is difficult to fabricate because it is geometrically complex. That is, the number of independent planes exceeds the company standard for spools.

<u>**Treatment Tank**</u>

Spool too large for treatment tank

A spool that requires galvanization or other tank treatment is too large for the tank.

No treatment tank found for <treatment_name> required by pipe spool

No treatment tank has been given in PipeBenders.xml for the treatment type called out by the pipe specification. This is an error in the editing of the PipeBenders.xml file, not a design error in the piping.

Individual pipes in spool require different tank treatments

A pipe spool is composed of multiple pipes that reference different pipe specs, and those pipe specs call out different, conflicting tank treatments.

<u>**Flange Welding**</u>

Flange hits machine or floor during bending, weld after bending

Flanges cannot be welded on the pipe before bending because they would hit the bending machine or the shop floor during bending. The simulator will have evaluated bending the pipe starting from either end before reporting this error, and will have checked all available bending machines with bend dies of the right size.

Pipe too long for automatic flange welding

This is a warning that a straight pipe with flanges on one or both ends is too long to be put through the automatic flange welder, and must be manually welded.

Pipe too short for automatic flange welding

This is a warning that a straight pipe with flanges on one or both ends is too short to be put through the automatic flange welder, and must be manually welded.

<u>**Inside Grinding and Painting**</u>

Pipe too long for inside coating

A pipe requires internal coating, but the pipe is too long for the available Pipe Internal Sprayer.

No internal sprayer found for inside coating

A pipe requires internal coating (as defined in the pipe spec entry in the XML file), but there is no Pipe Internal Sprayer available for the required coating material or for the pipe diameter.

Inaccessible for internal coating

A pipe that requires internal coating after bending has two or more bends. The sections between the bends are inaccessible for coating.

Bend on branch not allowed

A branch weld is inaccessible for grinding after welding because of a bend in the branch pipe.

Branch too far from end of main

Grinding cannot be done on a branch connection because it is too far from the end of the pipe.

Planning Inconsistencies Solution Messages

<u>**First Meet Check**</u>

First Meet assembly does not match the Planning Joint assembly

The planning joint is not located under the assembly containing the parts joined by the planning joint.

<u>**Hierarchy Check**</u>

Planning joint is not in an assembly that contains the two joined parts

The planning joint is located neither under the assembly containing the parts joined by the planning joint nor under a parent assembly/block of the assembly.

<u>**Physical Connection Check**</u>

Weld name is unknown

The weld bevel of a physical connection is not known because of a change to its associated planning joint.

<u>**Production Equipment Check**</u>

Production Equipment is not assigned to the assembly

The assembly has no production equipment assigned or the assigned production equipment is not compatible and cannot perform the weld.

<u>**Weld Side Check**</u>

The Weld Side of the Planning Joint is incorrect

The weld side of the planning joint is incorrect.

Property Dialog Boxes

This appendix contains reference information for all the property dialog boxes in the Structural Detailing task.

Common Property Tabs

The software displays some common property tabs on the properties dialog boxes for all Structural Detailing objects. Instead of repeatedly listing the common tabs with each object property dialog box, they are documented here for easy reference.

Relationship Tab (on page 400)
Members Tab (on page 401)
Controlled Interfaces Tab (on page 401)
Configuration Tab (on page 401)
General Tab (on page 402)
Parameters Tab (on page 402)

Relationship Tab

Displays all objects related to the selected object for which you are viewing properties. For example, if you are viewing the properties of a pipe run, the related pipeline, features, parts, associated control points, hangers or supports, and equipment display on this tab. All WBS assignments, including project relationships, appear on this tab.

Additional examples for marine relationships are as follows:

- For plate and profile system properties, the related bounded objects, bounding objects, and connections are shown.

- For plate and profile system part properties, parent systems are shown.

- For assembly connection properties, all connected objects are shown.

- For the properties of a frame connection on a member, supported, supporting, and auxiliary supporting parts are shown.

- For split connection properties, the parent and auxiliary supporting parts are shown.

Name

Specifies the name of the object.

Type

Specifies the type of object. To change the options on the list, edit the **Weld Type** select list in Catalog.

Go To

Displays the properties of the selected object.

Members Tab

Displays a grid containing the member (or child) objects of the smart occurrence object.

Name

Displays the name of the child object.

Type

The final result of rules for the smart occurrence, as displayed on the **Selection** tab. The result is used to create the child object.

Go To

Select a child object in the grid and click **Go To** to highlight the child object in a graphic view and in the **Workspace Explorer**.

Controlled Interfaces Tab

Displays the object properties and parameters that are created as a result of the smart occurrence rules.

Object

Displays the name of the object created by the smart occurrence rule. An object may appear more than once if multiple properties are created.

Interface

Display the name of the property or parameter created by the smart occurrence rule.

Rule Based

Displays a checkbox indicating that the property or parameter value is rule-based. Clear the check box to modify the value in on the appropriate tab of the **Properties** dialog box.

Configuration Tab

Displays the creation, modification, and status information about an object.

NOTE You cannot define the filters using the **Configuration** tab.

Plant

Displays the name of the model. You cannot change this value.

Permission Group

Specifies the permission group to which the object belongs. You can select another permission group, if needed. Permission groups are created in Project Management.

Transfer

Reassigns ownership of the selected model objects from their current permission group to another satellite or host permission group. This option is only available if the active model or project is replicated in a workshare configuration. The option is not available if all of the objects in the select set already belong to another location and are non-transferable. For more information, see *Transfer Ownership Dialog Box* in the *Common User's Guide.*

NOTE The **Transfer** option does not apply to the filters and surface style rules.

Approval State

Specifies the current status of the selected object or filter. The display depends on your access level. You might be unable to change the status of the object. The list is defined by the ApprovalStatus codelist.

🗒 **NOTE** You can only edit or manipulate an object with a status of **Working**.

Status

Specifies the location of the object in the workflow process. Changing this property sets the **Approval State**. The list is controlled by the ApprovalReason codelist in the ApprovalReason.xls file. You must bulkload this file. For more information, see *ApprovalReason* in the *Reference Data Guide*.

Date Created

Specifies the creation date of the object.

Created by

Specifies the name of the person who created the object.

Date Last Modified

Specifies the date when the object was last modified.

Last Modified by

Specifies the name of the person who last modified the object.

General Tab

The **General** tab displays the properties that were selected by you or automatically determined by the software at creation. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

The properties displayed duplicate those available on other tabs, such as the **Main** tab. You should edit the properties on these tabs.

Parameters Tab

Specifies the parameters for the smart occurrence object.

Parameter

Displays the name of the parameter.

Value

Displays the value of the parameter. Values that are grayed cannot be modified.

Rule-Based

Displays a check box if the parameter value is rule-based. Clear the check box to modify the value in the **Value** box.

Catalog Value

Displays the original catalog value if the **Value** box has been modified.

Smart Occurrence Ribbon

Displays the controls used to modify a smart occurrence object. The ribbon is available after the smart occurrence object is created automatically by the **Execute Detailing** command.

Properties

Activates the **Properties** dialog box, which you use to view and modify the properties of the smart occurrence object that you are modifying. The initial properties default from the Structural Detailing rules used by the **Execute Detailing** command.

Item

Displays the final result of User Answers on the **Selection** tab of the **Properties** dialog box.

NOTE **Item** does not display for some smart occurrence objects.

Plate Part Properties Dialog Box

Specifies the properties for the plate part that you are editing.

Main Tab (Plate Part Properties Dialog Box) (on page 403)
Material Tab (Plate Part Properties Dialog Box) (on page 405)
Relationship Tab (on page 400)
Configuration Tab (on page 401)
General Tab (Plate Part Properties Dialog Box) (on page 405)
Weight & CG Tab (on page 407)
Extended User Attributes Tab (on page 407)
Routing Tab (on page 407)

Main Tab (Plate Part Properties Dialog Box)

Specifies the general properties of the plate part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the object.

PlatePartRule - Creates a name based on the parent root plate system name, the location within the parent root plate system, and the parent assembly name.

User Defined - Name the plate part yourself by using the appropriate box.

Type

Displays the type of the plate part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the naming category. The naming rule uses the category in naming the profile part that is a child to the profile system.

Parent System

Displays the parent system for the object.

Surface Geometry Type

Displays the Molded Forms command used to create the root parent plate system. It is inherited from the parent system.

Specification

Specifies the specification for the object. This property is inherited from the parent system.

Description

Specifies a description for the object.

Tightness

Specifies the water tightness of the object. It is inherited from the parent system.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Material Tab (Plate Part Properties Dialog Box)

Displays the material properties for the plate part that you are editing. The material properties are inherited from the parent system.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Actual Thickness

Displays the material thickness for the object.

General Tab (Plate Part Properties Dialog Box)

Specifies general properties of the plate part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Plate Part Properties Dialog Box)* (on page 403).

PlateTightness

Specifies the water tightness of the plate part. The list is defined by the StructPlateTightness codelist.

PlateType

Specifies the type of the plate part. The list is defined by the StructPlateType codelist.

Thickening Technique Requested

Specifies the technique requested to thicken the plate part.

This property affects only the behavior of plate parts derived from systems. The property defaults to **Procedural** at creation. Changing the value of this property can affect the geometry of the part as described below.

Standalone plate parts initially display nothing for this property. You can change the value, but those changes do not change the part or its geometry. Smart 3D thickens all standalone plate parts, including collars, using procedural thickening.

Undefined (blank) indicates that Smart 3D selects the appropriate technique. Smart 3D automatically changes this value to **Procedural**.
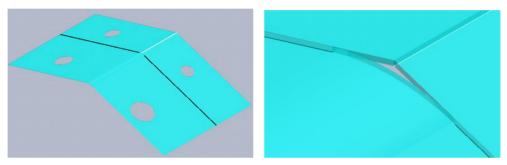
Procedural indicates the part is thickened using the standard thickening method provided by the ACIS modeler. This works for the vast majority of parts, and you should not change this value unless there is a specific reason to try a different technique.

📝 NOTE If the thickening fails using ACIS, and the part is a hull part, the software automatically tries the **NonProcedural** thickening technique. This option only controls which method the software tries first. You can determine which method Smart 3D actually used

from the **Thickening Technique Used** property.

Procedural with slit is used for parts that have internal slits or discontinuities in the surface of the system that form part of the boundary of parts on the system. The currently-supported example of this is a surface with a combination of split and bent knuckles joined end to end. Because Smart 3D does not support splitting knuckles and changing the properties of different segments, these surface must be constructed with the correct geometry to reflect the bent and split portions. Parts that are bounded by the slit, and potentially parts on the system that are within half a meter of the slit, must have this value set to assure a proper trim. Setting this attribute on parts that do not need it should not cause a failure but may cause the part to take more time to detail. The following pictures show an example of this type of surface, first the full surface and then a close-up in the region of the slit or discontinuity.

The plate system in these pictures is split into three parts by the knuckle (shown as white) and the two black seams. You must be sure that a single part does not touch both sides of the slit.

NonProcedural is an infrequently used option. Historically, a much higher percentage of parts failed to thicken with the ACIS modeler, and many designers used a specific hull form definition tool that produced a surface with very specific characteristics. **NonProcedural** thickening provided a tool that would thicken almost any part on this type of surface, even if the ACIS modeler failed. The ACIS modeler has improved greatly, and the **NonProcedural** thickening method has not been enhanced to handle the surfaces produced by many of the hull form tools currently in use. Setting this option may solve a failure-to-thicken problem on a hull part.

NOTE If **NonProcedural** thickening fails for a hull shell part, Smart 3D automatically tries to thicken the part using **Procedural** thickening. This option only controls which method the software tries first. You can determine which method Smart 3D actually used from the **Thickening Technique Used** property.

Thickening Technique Used

Displays the technique used to thicken the plate part. This is blank for a standalone plate part, although **Procedural** is always used for these parts. It displays one of the values defined for **Thickening Technique Requested** for a system-derived part. The meaning for these values is the same as defined for **Thickening Technique Requested**.

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the

BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Weight & CG Tab

Displays the center-of-gravity and the weight of the part.

Type

Displays the type of weight: **Dry Weight** or **Wet Weight**.

Weight

Displays the weight of the plate part pertaining to the weight type.

CoG X

Displays the position of center of gravity along the X-axis in the **CoG Coordinate System**.

CoG Y

Displays the position of center of gravity along the Y-axis in the **CoG Coordinate System**.

CoG Z

Displays the position of center of gravity along the Z-axis in the **CoG Coordinate System**.

CoG Coordinate System

Specifies the coordinate system to use to measure the center of gravity.

Extended User Attributes Tab

Displays the user attributes that have been bulkloaded on the report data object related to the part. Changes to these attributes do not cause the related manufacturing part to go out-of-date. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Routing Tab

Specifies routing properties for the part. By default, no routing properties are set.

Create

Sets routing properties for the part.

Delete

Deletes the routing properties for the part.

Rule

Specifies a predefined rule set or a user-defined rule that is applicable to the part.

Workcenter

Select an assembly workcenter where the part is produced from a facility-defined rule set. The list contains all available workcenters regardless of their level in the workcenter hierarchy. Examples of workcenters include docks, shops, bays, and panel lines. Workcenters may also be areas and zones where a block is assembled from multiple assemblies.

Stage Code

Specifies the workcenter code.

No. Of Actions

Specifies the number of actions required to produce the part.

Action

Specifies the name of the action.

Machine

Specifies the type of machine used to perform the required operation.

Code

Specifies the machine code.

Stiffener Part Properties Dialog Box

Specifies the properties for the stiffener part that you are editing. Default properties are inherited from the parent system of the part.

Main Tab (Stiffener Part Properties Dialog Box) (on page 408)

Profile Section Tab (Stiffener Part Properties Dialog Box) (on page 409)

Relationship Tab (on page 400)

Configuration Tab (on page 401)

General Tab (Stiffener Part Properties Dialog Box) (on page 410)

Weight & CG Tab (on page 407)

Extended User Attributes Tab (on page 407)

Routing Tab (on page 407)

Main Tab (Stiffener Part Properties Dialog Box)

Specifies the general properties of the stiffener part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide.*

Rule

Specifies the naming rule to use to name the stiffener part.

- Select **StiffenerPartRule** to create a name based on the parent root profile system name, the location within the parent root profile system, and the parent assembly name.

- Select **User Defined** to name the stiffener part using the appropriate box.

Type

Displays the type of the part. It is inherited from the parent system.

Subtype

Specifies an additional plate type that is independent of the **Type** value. The subtype does not affect molded conventions or plate naming. The default value is **None**.

Naming Category

Specifies the category for the stiffener part. Categories specify the role of the part in the model.

Parent System

Displays the parent system for the stiffener part.

Landing Curve Definition Method

Displays the landing curve definition.

Specification

Displays the structural specification for the stiffener part. It is inherited from the parent system.

Description

Type a description for the stiffener part.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Profile Section Tab (Stiffener Part Properties Dialog Box)

Specifies the profile section settings for the stiffener part.

Material

Displays the material for the stiffener part.

Grade

Displays the material grade for the stiffener part.

Section Type

Displays the profile section type.

Section Size

Displays the profile section size.

General Tab (Stiffener Part Properties Dialog Box)

Specifies general properties of the stiffener part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

ProfileType

Specifies the profile type of the part. The list is defined by the StrMfgProfileType codelist.

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Stiffener Part Properties Dialog Box)* (on page 408).

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

Assembly Connection Properties Dialog Box

Specifies the properties for the assembly connection that you are editing. Assembly connection properties affect the available selection of other smart occurrence objects that are children of the assembly connection.

Main Tab (on page 411)

Selection Tab (Assembly Connection Properties Dialog Box) (on page 411)

Relationship Tab (on page 400)

Controlled Interfaces Tab (on page 401)

Configuration Tab (on page 401)

Main Tab

Specifies the general properties of the smart occurrence object.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the **Name** box.

Parent System

Specifies the parent system for the object.

Type

Specifies the type of smart occurrence object.

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

♥ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not

changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

⊟ NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The available questions, answers, and results vary depending upon the detailed parts associated with the connection.

★ IMPORTANT The questions in each **Selection** tab represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Assembly Connection Properties Dialog Box (on page 410)
Smart Occurrence Ribbon (on page 403)

Slot Properties Dialog Box

Specifies the properties for the slot that you are editing.

See Also
Smart Occurrence Ribbon (on page 403)
Main Tab (on page 412)
Selection Tab (Slot Properties Dialog Box) (on page 413)
Parameters Tab (on page 402)
Members Tab (on page 401)
Controlled Interfaces Tab (on page 401)
General Tab (on page 402)

Main Tab

Specifies the general properties of the smart occurrence object.

Feature Name

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the

Name box.

Selection Tab (Slot Properties Dialog Box)

Specifies the rule criteria used for the slot. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

♀ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  📝 **NOTE** If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Assy Method

Select the method used place the profile stiffener in the slot. The options are **Drop** and **Slide**.

Apply Treatment

Select **Yes** to apply an edge treatment to the unconnected edges of the slot. An edge treatment object is created as a child of the slot, and is listed on the **Member** tab. For more information, see *Edge Treatment Properties Dialog Box* (on page 169).

Results

Select the general slot class.

Clearance

Specify the clearance between the profile and the slot.

Results

Select the slot geometry.

★ IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Smart Occurrence Ribbon (on page 403)
Slot Properties Dialog Box (on page 412)

Collar Properties Dialog Box

Specifies the properties for the collar that you are editing.

See Also
Smart Occurrence Ribbon (on page 403)
Main Tab (Collar Properties Dialog Box) (on page 414)
Material Tab (Collar Properties Dialog Box) (on page 416)
Selection Tab (Collar Properties Dialog Box) (on page 416)
Parameters Tab (on page 402)
Members Tab (on page 401)
Controlled Interfaces Tab (on page 401)
General Tab (on page 402)

Main Tab (Collar Properties Dialog Box)

Specifies the general properties of the collar plate part.

Name

Specifies the name of the plate part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the plate part. Select **User Defined** to type a name in the **Name** box.

Type

Specifies the type of plate part that you are placing. The default for a collar part is **Collar**.

Naming Category

Select a category for the plate part. The category specifies the role of the part in the model and is used by the naming rule to name the plate part.

Parent System

Specifies the parent plate system for the part. You can change the parent system to a system defined in the Systems and Specifications task. When the collar is created, smart occurrence rules may use the property values for the parent plate system. When a parent plate system property value changes, the rules update the corresponding plate part value.

Surface Geometry Type

Displays **Planar Plate**, similar to the Molded Forms command used to create a planar plate system.

Specification

Select the structural specification for the plate part.

Description

Type a description for the plate part.

Tightness

Select the level of tightness as it applies to the entire plate part.

Board Management

Symmetry

Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Manually Override

Turn on this option to stop the Board Management Service from considering this part.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

Structural Priority

Specifies the priority assigned to the object. Structural priority groups and filters plates, such as is needed in Drawings and Reports. The list is defined by the StructuralMemberPriority codelist.

Primary is the default value for Molded Forms plate systems.

Secondary is the default value for Molded Forms bracket systems.

Tertiary is the default value for Structural Detailing parts, such as collars, standalone plate parts, lapped plate parts, bracket parts, and plate edge reinforcements. These parts do not have parent systems.

Material Tab (Collar Properties Dialog Box)

Specifies the material properties for the collar that you are editing.

Material

Select the material for the plate part.

Grade

Select the material grade for the plate part.

Actual Thickness

Select the material thickness for the plate part.

See Also
Collar Properties Dialog Box (on page 414)
Smart Occurrence Ribbon (on page 403)

Selection Tab (Collar Properties Dialog Box)

Specifies the rule criteria used for the collar.

♥ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Stress Level

Select the amount of stress expected to be transferred by the collar between the profile stiffener and the parent plate. The options are **High**, **Medium**, and **Low**.

Collar Creation Order

Displays **Primary**. This value cannot be changed.

Collar Side Of Plate

Select **NoFlip** to place the collar on the side of the parent plate system with the fewest panel stiffeners. Select **Flip** to place the collar on the side of the parent plate system with the most panel stiffeners. Select **Centered** to place the collar at the center of the parent plate thickness.

Results

Select the general collar class.

Results

Select the collar geometry.

IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Collar Properties Dialog Box (on page 414)
Smart Occurrence Ribbon (on page 403)

Free End Cut Properties Dialog Box

Specifies the properties for the profile stiffener free end cut. The software creates a free end cut at the end of a profile part that is not connected to another part. Free end cut properties affect the available selection of the flange cut and web cut that are children of the free end cut.

See Also

Smart Occurrence Ribbon (on page 403)
Main Tab (on page 411)
Selection Tab (Free End Cut Properties Dialog Box) (on page 418)
Members Tab (on page 401)
Controlled Interfaces Tab (on page 401)

Main Tab

Specifies the general properties of the smart occurrence object.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the **Name** box.

Parent System

Specifies the parent system for the object.

Type

Specifies the type of smart occurrence object.

Selection Tab (Free End Cut Properties Dialog Box)

Specifies the rule criteria used for the profile stiffener free end cut.

⦿ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

End Cut Type

Select the free end cut type. A free end cut type contains a flange cut and a web cut. Flange cut and web cut objects are created as children of the free end cut, and are listed on the **Member** tab.

Results

Displays the result of the end cut type selection. This is a default value and cannot be changed.

IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Free End Cut Properties Dialog Box (on page 418)
Smart Occurrence Ribbon (on page 403)

Web Cut Properties Dialog Box

Specifies the properties for the profile stiffener web cut.

A web cut is based on the end cut type of the assembly connection. The end cut type must be changed at the assembly connection level.

The following web cuts are available:

Welded

Places a welded web and flange cut.

Snip

Places a snipped web and flange cut.

Cutback

Places a web and flange cut that are straight (not snipped), but are cut back from the bounding object.

Bracketed

Places a bracket as a child of the assembly connection.

Main Tab

Specifies the general properties of the smart occurrence object.

Feature Name

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the **Name** box.

Selection Tab (Web Cut Properties Dialog Box)

Specifies the rule criteria used for the profile stiffener free end cut.

TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.
- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  ◼ NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

User Answers

Displays answers to questions asked by the rule class, and a list of **Results** that match all of the answers. The default answers are determined by the rule class.

End Cut Type

Displays the end cut type, as defined by the assembly connection. Do not change this value. If you need to change the end cut type, change it on the Selection tab for the parent assembly connection or parent free end cut of the flange cut. For more information, see *Selection Tab (Assembly Connection Properties Dialog Box)* (on page 411) and *Selection Tab (Free End Cut Properties Dialog Box)* (on page 418).

Results

Displays the result of the end cut type selection. This is a default value and cannot be changed.

★ IMPORTANT

- A question with a check box uses the default value in the rules when the check box is selected.

- The questions in the Selection tab represent the default rules delivered with the software. User-customized rules may have different questions.

Flange Cut Properties Dialog Box

Specifies the properties for the profile stiffener flange cut.

A flange cut corresponds to a web cut. Both are based on the end cut type of the assembly connection. The end cut type must be changed at the assembly connection level.

See Also
Smart Occurrence Ribbon (on page 403)
Main Tab (on page 412)
Selection Tab (Flange Cut Properties Dialog Box) (on page 422)
Parameters Tab (on page 402)
Members Tab (on page 401)
Controlled Interfaces Tab (on page 401)
General Tab (on page 402)

Main Tab

Specifies the general properties of the smart occurrence object.

Feature Name

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the **Name** box.

Selection Tab (Flange Cut Properties Dialog Box)

Specifies the rule criteria used for the profile stiffener flange cut.

♀ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  📝 **NOTE** If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.
2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

End Cut Type

Displays the end cut type, as defined by the assembly connection. Do not change this value. If you need to change the end cut type, change it on the **Selection** tab for the parent assembly connection or parent free end cut of the flange cut. For more information, see *Selection Tab (Assembly Connection Properties Dialog Box)* (on page 411) and *Selection Tab (Free End Cut Properties Dialog Box)* (on page 418).

Bottom Flange

Displays **No** if the profile does not have a bottom flange. Displays **Yes** if the profile has a bottom flange. The answer to **Bottom Flange** is determined by the assembly connection based on the cross-section of the profile, and should not be changed.

Apply Treatment

Select **Yes** to apply an edge treatment to the flange cut. An edge treatment object is created as a child of the flange cut, and is listed on the **Member** tab. For more information, see *Edge Treatment Properties Dialog Box* (on page 169).

Results

Select the flange cut geometry.

⭐ **IMPORTANT** The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also
Flange Cut Properties Dialog Box (on page 422)
Smart Occurrence Ribbon (on page 403)

Physical Connection Properties Dialog Box

Specifies the properties for the physical connection that you are editing.

See Also

Main Tab (Physical Connection Properties Dialog Box)

Specifies the general properties of the physical connection.

Name

Specifies the name of the physical connection. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the physical connection. Select **User Defined** to type a name in the **Name** box.

Joint

Method

Specifies the method of connection, such as **Welded**.

Type

Specifies the type of connection using the specified **Method**, such as **Tee Weld** or **Butt Weld**. **Type** is the same as **Class** on the **Selection** tab.

Connected Parts

Name

Specifies the name of the detailed parts connected by the physical connection. Each detailed part is listed separately.

See Also

Selection Tab (Physical Connection Properties Dialog Box)

Specifies the rule criteria used for the physical connection.

♥ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The available questions, answers, and results vary depending upon the type of physical connection.

★ **IMPORTANT** The questions in each **Selection** tab represent the default rules delivered with the software. User-customized rules may have different questions.

Bevel Parameters Tab

Specifies the bevel parameters used for the smart occurrence object, such as a physical connection or free edge treatment.

Reference Part

Specifies the part containing the bevel.

Depth

Specifies the depth of the bevels. The **Depth** boxes correspond to the dimensions on the preview picture.

Angle

Specifies the angle of the bevels. The **Angle** boxes correspond to the dimensions on the preview picture.

Parameters Tab (Physical Connection Properties Dialog Box)

Specifies the parameters for the smart occurrence object.

Parameter

Displays the name of the parameter.

Value

Displays the value of the parameter. Values that are grayed cannot be modified.

Rule-Based

Displays a check box if the parameter value is rule-based. Clear the check box to modify the value in the **Value** box.

Catalog Value
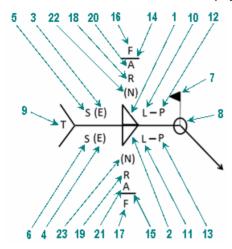
Displays the original catalog value if the **Value** box has been modified.

Welding Symbol Parameters

In addition to physical connection parameters, welding symbol parameters are also visible. Weld symbols are used by the Drawings and Reports task but welding symbol parameters are only visible on the **Parameters** tab for a physical connection:

1 - Primary Side Symbol or Primary Side Groove

13 - Secondary Side Pitch or Distance Between the Ends of Adjacent Welds

2 - Secondary Side Symbol or Secondary Side Groove

14 – Primary Side Contour

3 – Primary Side Groove Size

15 – Secondary Side Contour

4 – Secondary Side Groove Size

16 – Primary Side Finish Method

5 - Primary Side Bevel Depth or Leg Length or Actual Throat Thickness or Strength

17 – Secondary Side Finish Method

6 - Secondary Side Bevel Depth or Leg Length or Actual Throat Thickness or Strength

18 – Primary Side Root Opening or Depth of Filling

7 – Field Weld

19 - Secondary Side Root Opening or Depth of Filling

8 - All Around

20 – Primary Side Groove Angle or Included Angle of Countersink

9 – Tail Notes

21 - Secondary Side Groove Angle or Included Angle of Countersink

10 – Primary Side Length

22 – Primary Side Number Of Welds

11 – Secondary Side Length

23 - Secondary Side Number Of Welds

12 – Primary Side Pitch or Distance Between the Ends of Adjacent Welds

The actual parameters provided depend upon the type of physical connection, but the following figures are representative. For a definition of the parameters, please refer to the physical connection documentation in the *Structural Detailing Reference Data Guide*.

The following example shows the welding symbol properties for a Y tee weld. Other tee welds and lap welds are similar.

PrimarySideSymbol	None	
SecondarySideSymbol	None	
PrimarySideGroove	None	☑
SecondarySideGroove	Bevel	☑
PrimarySideGrooveSize	0.0000 m	☑
SecondarySideGrooveSize	0.0000 m	☑
PrimarySideActualThroatThickness	0.0000 m	☑
SecondarySideActualThroatThickness	0.0090 m	☑
PrimarySideNominalThroatThickness	0.0000 m	☑
SecondarySideNominalThroatThickness	0.0000 m	☑
FieldWeld	0	

AllAround	0	
PrimarySideSupplementarySymbol	None	
SecondarySideSupplementarySymbol	None	
TailNotes		☑
TailNoteIsReference	0	☑
PrimarySideLength	0.0000 m	☑
SecondarySideLength	0.0000 m	☑
PrimarySidePitch	0.0000 m	☑
SecondarySidePitch	0.0000 m	☑
PrimarySideContour	None	
SecondarySideContour	None	

PrimarySideFinishMethod	None	
SecondarySideFinishMethod	None	
PrimarySideRootOpening	0.00 m	☑
SecondarySideRootOpening	0.00 m	
PrimarySideGrooveAngle	0.00 deg	☑
SecondarySideGrooveAngle	45.00 deg	☑
PrimarySideActualLegLength	0.00 m	☑
SecondarySideActualLegLength	0.00 m	☑

The next example shows the welding symbol parameters for a V butt weld. Fewer of the parameters are exposed because many are not applicable to a butt weld.

PrimarySideSymb	None	
SecondarySideS	None	
PrimarySideGroo	V	☑
SecondarySideG	None	☑
PrimarySideGroo	0.0000 m	☑
SecondarySideG	0.0000 m	☑
PrimarySideActua	0.0120 m	☑
SecondarySideA	0.0000 m	☑
FieldWeld	0	
PrimarySideSupp	None	
SecondarySideS	None	

Logical Connection Properties Dialog Box

Specifies the properties for logical connections.

Main Tab (Logical Connections Properties Dialog Box) (on page 429)

Relationships Tab (Logical Connections Properties Dialog Box)

Configuration Tab (on page 401)

General Tab (Logical Connections

Properties Dialog Box)

Main Tab (Logical Connections Properties Dialog Box)

Specifies the main properties for the logical connection.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Specifies the naming rule to use to name the logical connection.

- Select **StdHierarchyChildNamingRule** to use the following syntax: *<Parent Name>-*LC*<Index Number>*. For example, D0-1DCK-1-LC101, where D0 is the reference plane, 1DCK is an index number appended by the plate type (Deck in this example), 1 is the Workshare Location ID, and LC101 is "Logical Connection" appended by an index number.

- Select **User Defined** to name the logical connection yourself by using the appropriate box.

Parent System

Displays the name of the parent system of the logical connection.

Type

Displays the object type, which in this case is **Logical Connection**.

Optional Corner Trim Property

📄 **NOTE** Your administrator must bulkload the following files before the **Corner Trim** displays:

Bulkload into the site and site schema databases:

- *[Product Folder]*ShipCatalogData\BulkLoad\DataFiles\ProjectDBFiles\StructureCustomClasses.xls

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimCodelist.xls

- Bulkload into the catalog and catalog schema databases:

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\SM_SquareEdgeTrimRules.xls

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimCodelist.xls

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimAttribute.xls

Corner Trim

Indicates whether plates bounded by plate connections or edge reinforcement connections use the square edge trim.

The available options are:

- **Flush (Flush Neat Trim)** - This value uses the normal trim behavior for attached plates or for plates bounded by edge reinforcement when detailing. This is the default value.

- **Open (Open Square Trim)** - This value uses the square edge trim behavior when detailing the attached plates or when detailing the plates bounded by edge reinforcement, as shown below.

Plate bounded by plate:
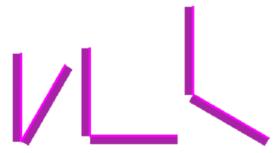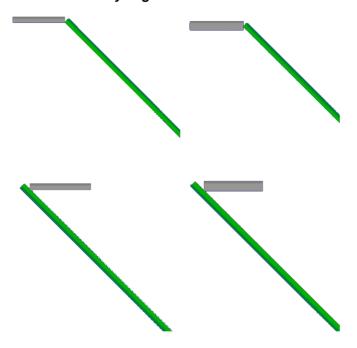
Plate bounded by edge reinforcement:

General Tab (Logical Connections Properties Dialog Box)

Displays the general properties for the logical connection.

Corner Trim

Indicates whether plates bounded by plate connections or edge reinforcement connections use the square edge trim. You must bulkload the following files before this option displays:

The following files need to be loaded into the site and site schema databases:

[Product Folder]\ShipCatalogData\BulkLoad\DataFiles\ProjectDBFiles\StructureCustomClasses.xls

[Product Folder]\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimCodelist.xls

The following files need to be loaded into the catalog and catalog schema databases:

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\SM_SquareEdgeTrimRules.xls

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimCodelist.xls

- *[Product Folder]*\ShipCatalogData\Bulkload\DataFiles\ShipStructure\CornerTrimAttribute.xls

The available options are:
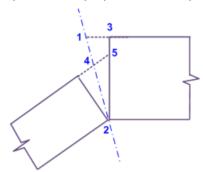
Flush (Flush Neat Trim)

This value uses the normal trim behavior for attached plates or for plates bounded by edge reinforcement when detailing. This is the default value.

Open (Open Square Trim)

> This value uses the square edge trim behavior when detailing the attached plates or when detailing the plates bounded by edge reinforcement

Chamfer Properties Dialog Box
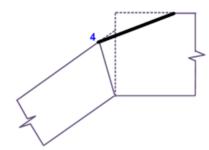
Specifies the properties for the plate chamfer that you are editing.

On planar plates parts, a chamfer is based on the distance between the outer surfaces of the two parts (line 1-4 and line 3-5, which are equal for planar plates).

At a plate knuckle, the plate parts are split is along the bisecting angle of the knuckle (line 1-2). A chamfer is based on the distance between the outer surfaces of the two parts, measured from the split location (4) to the outer surface of the thicker plate (line 1-3). The chamfer location begins at the actual split location (4):
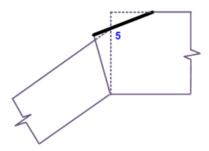
Chamfers on Offshore Structure

In some cases, chamfers are treated differently for offshore structure.

For cans, used in tubular offshore structure:
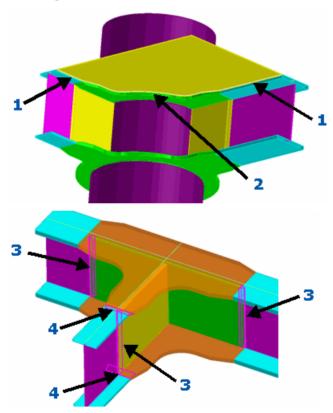
The chamfer is based on the distance between the outer surfaces of the two plate parts, measured long the normal edge of the tube (line 3-2) from the "theoretical" split location (5) to the outer surface of the thicker plate (3). The chamfer location passes through the "theoretical" split location (5):

For other offshore structure:

1. Chamfers are not placed on the length edge of a built-up flange.

2. Chamfers are not placed on the edge of a side plate or ring plate that is bounding the secondary deck plate

3. Chamfers are placed on continuity and transition plates when they are butt welded to the web of a member.

4. Chamfers are placed on the edge of a side plate and ring plate that is butt welded to the flange of a member.

See Also
Smart Occurrence Ribbon (on page 403)
Main Tab (on page 412)
Selection Tab (Chamfer Properties Dialog Box) (on page 434)
Parameters Tab (on page 402)

Members Tab (on page 401)
Controlled Interfaces Tab (on page 401)

Main Tab

Specifies the general properties of the smart occurrence object.

Feature Name

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use to name the object. Select **User Defined** to type a name in the **Name** box.

Selection Tab (Chamfer Properties Dialog Box)

Specifies the rule criteria used for the plate chamfer.

⬤ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

Questions, Answers, and Results

The default answers are determined by the rule class.

Chamfer Type

Displays the type of chamfer, as defined by the assembly connection. Do not change this value.

Chamfer Weld

Displays the weld type used by the chamfer, as defined by the assembly connection. Do not change this value.

Results

Select the general chamfer class.

Results

Select the chamfer geometry.

IMPORTANT The questions described here represent the default rules delivered with the software. User-customized rules may have different questions.

See Also

Chamfer Properties Dialog Box (on page 432)
Smart Occurrence Ribbon (on page 403)

Linear Beam Properties Dialog Box

Specifies the properties for the linear beam.

See Also

Cross Section Tab (Linear Beam Properties Dialog Box) (on page 437)
Main Tab (Linear Beam Properties Dialog Box) (on page 436)
Relationship Tab (on page 400)
Section Orientation Tab (Linear Beam Properties Dialog Box) (on page 437)

Main Tab (Linear Beam Properties Dialog Box)

Specifies the general properties for the linear beam part.

Name

Displays the name to assign to the beam part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Specifies how the name is created. If set to a naming rule, the software automatically generates the name for you. If you select **User Defined**, you can type in the linear beam name in the **Name** box.

Type

Select the type of linear beam.

Naming Category

Select a naming category. The category is used by the naming rule to name the profile part that is a child to the profile system.

Parent System

Displays the name of the parent system of the linear beam.

Specification

Select a specification for the edge reinforcement linear beam.

Description

Type a description for the linear beam.

Board Management

Symmetry

Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

Cross Section Tab (Linear Beam Properties Dialog Box)

Specifies the cross section properties for the linear beam.

Section Type

Select the profile section type. The software displays the section type properties for the section type that you selected. Each section type has different properties.

Section Size

Select the profile section size.

Material

Select the material for the profile.

Grade

Select the material grade for the profile. Different material grade are available depending on the material that you selected.

See Also
Linear Beam Properties Dialog Box (on page 436)

Section Orientation Tab (Linear Beam Properties Dialog Box)

Specifies the position of the linear beam cross section.

Beam Axis Orientation Rule

Specifies the direction to which the linear beam axis is constrained.

Beam Type

Specifies the type of linear beam.

Primary Orientation

Specifies the primary orientation of the linear beam's cross section. The local v-axis of the cross section is aligned with this direction.

Load Point

Select the cross-sectional load point through which the linear beam axis will pass.

See Also
Linear Beam Properties Dialog Box (on page 436)

Beam Part Properties Dialog Box

Specifies the properties for the beam part that you are editing. Default properties are inherited from the parent system of the part.

Main Tab (Beam Part Properties Dialog Box) (on page 438)

Cross Section Tab (on page 443)

Section Orientation Tab (Beam Part Properties Dialog Box) (on page 439)

Relationship Tab (on page 400)

Configuration Tab (on page 401)

General Tab (Beam Part Properties Dialog Box) (on page 439)

Weight & CG Tab (on page 407)

Extended User Attributes Tab (on page 407)

Main Tab (Beam Part Properties Dialog Box)

Specifies the general properties of the beam part.

Name

Specifies the name of the object. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the beam part.

- Select **BeamPartRule** to create a name based on the parent root profile system name, the location within the parent root profile system, and the parent assembly name.

- Select **User Defined** to name the part yourself by using the appropriate box.

Type

Displays the type of the part. It is inherited from the parent system.

Naming Category

Select a category for the beam part. Categories specify the role of the part in the model.

Parent System

Displays the parent system for the beam part.

Specification

Displays the structural specification for the beam part. It is inherited from the parent system.

Description

Type a description for the beam part.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Section Orientation Tab (Beam Part Properties Dialog Box)

Specifies the position of the linear beam cross-section. The properties duplicate those available on the **Section Orientation** tab in the **Linear Beam Properties** dialog box.

Beam Axis Orientation Rule

Specifies the direction to which the linear beam axis is constrained.

Beam Type

Specifies the type of linear beam.

Primary Orientation

Specifies the primary orientation of the linear beam's cross-section. The local v-axis of the cross-section is aligned with this direction.

Load Point

Select the cross-section load point through which the linear beam axis passes.

See Also
Beam Part Properties Dialog Box (on page 438)

General Tab (Beam Part Properties Dialog Box)

Specifies general properties of the stiffener part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

Category: Standard

BeamAxisRuleType

Specifies the direction to which the linear beam axis is constrained.

BeamType

Specifies the type of linear beam.

Description

Specifies a description for the object.

FlipAboutU

Specifies whether to flip the linear beam cross-section about the local u-axis of the beam.

FlipAboutV

Specifies whether to flip the linear beam cross-section about the local v-axis of the beam.

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Beam Part Properties Dialog Box)* (on page 438).

PrimeOrientation

Specifies the primary orientation of the linear beam's cross-section. The local v-axis of the cross-section is aligned with this direction.

RotationAngle

Specifies the rotation of the cross-section about its local axis.

Category: Planning

Build Method

Specifies the method used to position child objects in the block. The list contains all available values as defined in the reference data for the assigned workcenter, typically **Vertical Drop**, **Drop at angle**, **Slide**, and **Default**. The list is defined by the BuildMethodData codelist.

Slot Connectivity

Specifies the minimum welding requirement at profile and slot intersections to meet the build method requirements. The list contains all available values as defined in the reference data, typically **None**, **Webleft**, **Webright**, **Double**, **N/A**, and **Default**. The list is defined by the SlotConnectivityData codelist.

See Also
Beam Part Properties Dialog Box (on page 438)

Plate Edge Reinforcement Part Properties Dialog Box

Specifies the properties for the plate edge reinforcement that you are editing.

See Also
Main Tab (Plate Edge Reinforcement Part Properties Dialog Box) (on page 441)
Material Tab (on page 100)
Configuration Tab (on page 401)

Main Tab (Plate Edge Reinforcement Part Properties Dialog Box)

Specifies the general properties of the standalone plate edge reinforcement part.

Name

Displays the name that will be assigned to the part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Specifies how the name is created. If set to a naming rule, the software automatically generates the name for you. If you select **User Defined**, you can type in the part name yourself in the **Name** box.

Type

Displays the object type, which in this case is Edge Reinforcement.

Naming Category

Select a category for the part. Categories specify the role of the part in the model. The category is also used by the naming rule to name the part.

Parent System

Displays the name of the parent system of the part.

Specification

Select the structural specification for the plate system.

Description

Type a description for the plate system.

Tightness

Select the level of tightness as it applies to the entire plate system.

Symmetry

Displays the symmetry value for the part. The symmetry value is determined by the Board Management Service.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by the Board Management Service.

Material Tab

Specifies the material properties for the plate system that you are editing.

Material

Specifies the object material type, such as **Steel - Carbon** or **Steel - High Strength**.

Grade

Specifies the object material grade, such as **A36** or **A529**.

Thickness

Specifies the material thickness for the plate system.

See Also
Place Plate Parts Properties Dialog Box (on page 98)
Plate Edge Reinforcement Part Properties Dialog Box (on page 441)

Edge Reinforcement Profile Part Properties Dialog Box

Specifies the properties for the edge reinforcement profile part that you are editing. Default properties are inherited from the parent system of the part.

Main Tab (Edge Reinforcement Profile Part Properties Dialog Box) (on page 442)

Cross Section Tab (on page 443)

Section Orientation Tab (Edge Reinforcement Profile Part Properties Dialog Box) (on page 444)

Relationship Tab (on page 400)

Configuration Tab (on page 401)

General Tab (Edge Reinforcement Profile Part Properties Dialog Box) (on page 444)

Weight & CG Tab (on page 407)

Extended User Attributes Tab (on page 407)

Main Tab (Edge Reinforcement Profile Part Properties Dialog Box)

Specifies the general properties of the edge reinforcement profile part.

Name

Specifies the name of the edge reinforcement profile part. Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the *Global Workshare Guide*.

Rule

Select the naming rule to use to name the edge reinforcement profile part.

- Select **StiffenerPartRule** to create a name based on the parent root profile system name, the location within the parent root profile system, and the parent assembly name.

- Select **User Defined** to name the part yourself by using the appropriate box.

Type

Displays the type of the part. It is inherited from the parent system.

Naming Category

Select a category for the edge reinforcement profile part. Categories specify the role of the part in the model.

Parent System

Displays the parent system for the edge reinforcement profile part.

Specification

Displays the structural specification for the edge reinforcement profile part. It is inherited from the parent system.

Description

Type a description for the edge reinforcement profile part.

Board Management

Symmetry

Specifies the symmetry value for the part. The symmetry value is determined by **Tools** > **Board Management Service** in Structural Detailing.

Manually Override

When selected, allows you to manually change the **Symmetry** value.

Symmetrical Part

Displays the name of the symmetrical part, if available. The symmetrical part is determined by **Tools** > **Board Management Service** in Structural Detailing.

Cross Section Tab

Specifies the cross-section settings for the part.

Section Type

Displays the profile section type.

Section Size

Displays the profile section size.

Material

Displays the material for the part.

Grade

Displays the material grade for the part.

 NOTE The tab also displays the graphic and parameters of the cross-section symbol used by the part.

Section Orientation Tab (Edge Reinforcement Profile Part Properties Dialog Box)

Specifies the section orientation settings for the edge reinforcement profile part.

Position

Displays how the cross-section of the profile is attached to the free edge.

Primary Orientation

Displays which side of the plate to place the edge reinforcement profile.

Secondary Orientation

Displays the direction of the web thickness for symmetrical crossing-sections and the direction of the flange and web thickness for unsymmetrical cross-sections.

Parent System Type

Displays the object type of the parent system.

Offset

Displays the distance that the cross-section is offset from free edge or from the plate face.

Mounting Face

Displays the cross-section face to mount to the plate.

Load Point

Displays the cross-section load point through which the landing curve passes.

General Tab (Edge Reinforcement Profile Part Properties Dialog Box)

Specifies general properties of the stiffener part. Some properties displayed duplicate those available on other tabs, such as the **Main** tab. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid.

flangeDirection

Specifies the direction of the web thickness for symmetrical crossing-sections or the direction of the flange and web thickness for unsymmetrical cross-sections.

NamingCategory

Displays the codelist value for **Naming Category**, defined on the **Main** tab. For more information, see *Main Tab (Edge Reinforcement Profile Part Properties Dialog Box)* (on page 442).

orientation

Specifies the orientation of the edge reinforcement.

ProfileType

Specifies the profile type of the part. The list is defined by the StrMfgProfileType codelist.

webThicknessOffset

Specifies the offset direction along the plate face.

---

webThicknessOrientation

This property is not used.

Sketched Feature Properties Dialog Box

Specifies the properties for the sketched feature.

See Also
Main Tab (Sketched Feature Properties Dialog Box) (on page 445)
Catalog Tab (Sketched Feature Properties Dialog Box) (on page 447)
Configuration Tab (on page 401)

Main Tab (Sketched Feature Properties Dialog Box)

Feature Name

Name

Specifies the name of the sketched feature.   Names generated by a rule include a Global Workshare name rule ID if the name rule ID was defined when the model database was created. For more information, see *Using Global Workshare* in the Global Workshare Guide.

Rule

Select the naming rule to use. Select **User Defined** to specify the name yourself in the **Name** box.

Opening design type

Specifies the design opening property for the opening or hole. The default value is **Permanent Access**. The list is defined by the CutoutDesignType codelist.

Opening planning method

Specifies the planning opening property for the opening or hole. The default value is **Design**. The list is defined by the CutoutPlanningMethod codelist.

Opening production method

Specifies the production opening property for the opening or hole. The default value is **Cut**. The list is defined by the CutoutProductionMethod codelist.

▤ NOTE For more information about sketched feature opening properties, see *Place Sketched Features* (on page 160).

Board Management

The **Tools** > **Board Management Service** command determines structural part and seam symmetry about the center plane.

- Naming rules can consider symmetry information. For example, you can include a suffix indicating the symmetry property value for parts. Naming rules can also sort parts within a block by the symmetry value before assigning a part name index.

- The board property can be used in the nesting and cutting processes by possibly nesting and then cutting symmetrical parts at the same time, or nesting the symmetrical parts on the same raw material stock if the parts are in the same block. The Board Management Service

determines if parts are symmetrically similar. Structural Manufacturing determines if the parts are symmetrically identical for manufacturing purposes.

- Unfolding algorithms use seam symmetry information when they calculate shapes for plates that cross the centerline.

- Some assembly planning tasks use the board property to automate creating and maintenance of assembly hierarchies for symmetrical parts.

- Reports include part symmetry information.

- Seam board information supports shell expansion drawing requirements. The shell expansion drawing for the entire ship displays and labels both port and starboard shell seams in one view.

- You can choose to manually override board management review for an object. If you choose this option, then board management disregards the object for subsequent review.

Whether you choose to run the Board Management service periodically or continuously depends upon your processes and workflow. If you need to see the symmetry immediately, then running this process continuously on a remote server may best suite your needs. If you do not need immediate results, then you can run it periodically on your local computer at times that best fit your workflow.

NOTE This property affects how Smart 3D names parts. It also affects how the parts are handled in Planning and Structural Manufacturing. Because of this, we recommend that you run the Board Management service before you generate drawings, bills of materials, or manufacturing output.

Board Management Service Console

Specifies the settings for running the board management service.

Server

Displays the name of the server on which the board management service runs.

Started by

Displays the user name of the person who started the board management service.

Seams Selected for Evaluation

Hull Seams

Indicates that seams on hull surfaces should be evaluated for symmetry.

Interior Seams

Indicates that seams on interior surfaces should be evaluated for symmetry.

Filters Applied to Target Selection

by Type

Indicates that parts should be filtered by type. The **Type** property is set on the **System Properties** dialog box.

by Naming Category

Indicates that parts should be filtered by naming category. The **Naming Category** property is set on the **System Properties** or **Part Properties** dialog box.

by Region

Indicates that parts should be filtered by region or bounding box. The bounding box is defined by the minimum and maximum x, y, and z locations of the object.

Growth

Specifies the amount to expand or shrink the region before filtering. The intent is to find parts that are approximately the same size and in approximately the same (mirrored) location.

Previous Part List Date/Time

Displays the date and time of the previous generation of the list of new and modified parts.

Reset Date/Time to

Replaces the start value of the last time the service was run. This time is used to search for new and modified parts.

Processing Status

Displays information about the number of parts processed and an estimation of the time remaining.

Start

Starts the board management service.

Stop

Quits the board management service without completing the process.

Close

Saves the values and closes the dialog box.

See Also
Sketched Feature Properties Dialog Box (on page 445)

Catalog Tab (Sketched Feature Properties Dialog Box)

Lists all the sketched feature shapes that are available in the catalog. Using the tree view, browse through the catalog to find the feature. You can see a preview in the **Preview** area. All parameters and values for those parameters also display when you select a sketched feature opening from the catalog.

See Also
Sketched Feature Properties Dialog Box (on page 445)

Assembly Connection Properties Dialog Box

Specifies the properties for the member assembly connection that you are editing.

Occurrence Tab (Assembly Connection Properties Dialog Box) (on page 289)
Definition Tab (Assembly Connection Properties Dialog Box) (on page 293)
Relationship Tab (on page 400)
Configuration Tab (on page 401)
Notes Tab
Selection Tab (Assembly Connection Properties Dialog Box) (on page 294)

See Also
Edit assembly connection properties (on page 287)

Occurrence Tab (Assembly Connection Properties Dialog Box)

The **Occurrence** tab displays the member assembly connection properties that you can edit or that are automatically determined by the software at placement. The property name appears on the left side of the grid, and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection, and then selected the properties command, only the common properties between the selected assembly connections display.

When viewing properties for a single assembly connection, the following properties display. More properties may display depending on what you defined in the reference data. Refer to the Structure Reference Data Guide for more information on properties.

Name

Displays the name of the assembly connection. The assembly connection name is based on the **Name Rule** selection. To type a new name for the assembly connection, in the **Name Rule** box, select **User Defined**, and then type a name for the assembly connection in the **Name** box.

Name Rule

Specify the naming rule to use to name this assembly connection.

- **Default Name Rule** - Names the assembly connection using this format: <assembly connection>-<location>-<unique index> where <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, GussetPlateAsmConn_1-1-0045.

- **Unique Name Rule** - Names the assembly connection using this format: <parent system name>-<assembly connection>-<location>-<unique index> where <parent system name> is the parent System selected for the assembly connection, <assembly connection> is the name of the assembly connection defined in the catalog, <location> is the global workshare location, and <unique index> is an index number that starts at 0001. For example, Structure System-GussetPlateAsmConn_1-1-0045.

- **User Defined** - Select to specify the assembly connection name yourself in the **Name** box.

System

Select the system to which the assembly connection that you are placing belongs. You can

create new systems in the Systems and Specifications task.

Base Plate Assembly Connection Properties

Depth Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension C in the figure.

Width Clearance

Specify the clearance between the flange of the member and the edge of the base plate. This is dimension D in the figure.

Sizing Rule

Select the sizing rule method for the base plate.

Plate Category

Select the plate category.

Plate Type

Select the plate type.

Miter Assembly Connection Properties

Top Distance

Specifies the distance between the top flange of the member section and the top of the plate.

Bottom Distance

Specifies the distance between the bottom flange of the member section and the bottom of the plate.

Left Distance

Specifies the distance between the left edge of the member section and the left edge of the plate.

Right Distance

Specifies the distance between the right edge of the member section and the right edge of the plate.

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select

Center to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

With Plates

Specifies whether or not a plate is inserted between the member ends. Select **False** to not place the plate. Select **True** to place the plate.

Clearance

Specifies the distance between the member ends.

Sizing Rule

Select the sizing rule method for the base plate.

Slab by Member Boundary

Clearance

Type a clearance distance between the edge of the slab and the member.

Port Face Position

Select the location on the member at which the slab is to stop. You can select the outmost plane, the centerline, or the in-most plane on the member.

Detailed Connection

Select **True** to trim the slab using the boundary member as the cutting edge. Select **False** to not trim the slab.

Offset

Specify the distance between the selected **Port Face Position** and the edge of the slab. A negative value moves the edge into the body of the slab. A positive value moves the edge out from the body of the slab.

Slab by Slab Assembly

Trim Type

Defines the type of trimming.

- **By Whole Slab** - Trim by the slab boundary geometry before trim.
- **By Lateral Face** - Trim by the slab boundary geometry after trim.
- **By Extended Lateral Face** - Trim by the extension of the lateral face of the slab boundary.

Trim Excess Material

Specify whether to trim the exceeding part.

Trim Excess Material Offset

Specify the offset applied on the lateral face used to trim.

Slab Free Edge Assembly

Reference Direction

Select the reference direction for the angle:

Normal

The angle is measured from a vector perpendicular to the slab edge.

- ▪ **Horizontal** - The angle is measured from the global XY plane in the model.
- ▪ **Vertical** - The angle is measured from the Z-Axis in the model.

Angle

Type a slope for the slab edge represented by the assembly connection. If the slab was place using the **Face Position Top**, the side face rotates about the top slab edge. If the slab was placed using the **Face Position Bottom**, the side face rotates about the bottom slab edge.

Offset

Specify the distance between the selected boundary object and the edge of the slab.

Splice Assembly Connection Properties

Symmetry

Controls how to cut back the member ends when the clearance value is not zero. Select **Center** to specify that both members are cut back equally. Select **Right** to specify that the first member that you selected be cut back. Select **Left** to specify that the second member that you selected be cut back.

Clearance

Specifies the distance between the ends of the members.

Splice With

Select the plates that you want to use in the splice.

Web Plate Position

Select a web plate position.

Distance from flange gage line

Specifies the distance from the flange gage line.

Distance from web gage line

Specifies the distance from the web gage line.

Flange Plate Thickness

Specifies the thickness of the flange plates.

Flange Plate Length

Specifies the length of the flange plates.

Flange Plate Width

Specifies the width of the flange plates.

Flange Plate Category

Select the plate category for the flange plates.

Flange Plate Type

Select the plate type for the flange plates.

Web Plate Thickness

Specifies the thickness of the web plates

Web Plate Length

Specifies the length of the web plates.

Web Plate Width

Specifies the width of the web plates.

Web Plate Category

Select the plate category for the web plates.

Web Plate Type

Select the plate type for the web plates.

General Surface Assembly Connection Properties

With Pad

Select **True** to place a steel pad between the member end and the surface. Select **False** to have the member end connect directly to the surface.

Pad Type

Select the shape of the pad.

Sizing Rule

Select whether you want to the software to automatically size the pad or if you want to size the pad.

Offset

Type the distance between the end of the member and the face of the surface.

See Also
Assembly Connection Properties Dialog Box (on page 289)

Definition Tab (Assembly Connection Properties Dialog Box)

The **Definition** tab displays the member assembly connection properties as they are defined in the reference data. The property name appears on the left side of the grid and the corresponding property value appears on the right side of the grid. If you selected more than one assembly connection and then selected the properties command, only the common properties between the selected assembly connections display.

The properties that display depend on what you defined in the reference data. Refer to the *Structure Reference Data Guide* for more information on the properties.

See Also
Assembly Connection Properties Dialog Box (on page 289)

Selection Tab (Assembly Connection Properties Dialog Box)

Specifies the rule criteria used for the member assembly connection. Assembly connection rule selections affect the selections available for other smart occurrence objects that are children of the assembly connection.

♥ TIPS

- The software reevaluates the selection rules after any change occurs to the values, when you select a new object, or when you open a different property page.

- You can resize the columns by sliding the divider in the heading row.

Class

Displays the name of the rule class.

Question

Displays questions asked by the rule.

Answer/Result

Defines the results of the rules and the answers to questions asked by the rules. The default results and answers are determined by the rule class. Each row represents a result or answer that is cumulatively defined by the preceding rows. The **Answer/Result** box is not editable if **Rule Based** is selected.

Rule Based

Defines when a question is rule-driven. This option is selected by default for each question.

- Select to always use the default rule-based answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the answer is changed to the new default.

- Clear to manually select an answer or result. For an existing object, if a change in the model requires a recalculation of the rule, then the manually-selected answer is not changed to the new default. If the manually selected answer is not in the list of valid answers, then the new default is used.

  ▤ NOTE If multiple existing objects are selected and a row contains a mix of rule-based and non-rule-based answers/results, then the **Answer/Result** box is blank, and the **Rule Based** check is unavailable, as shown in the following example.

1. Click the **Rule Based** box to clear and remove the rule-based value for all objects.

2. Click the **Rule Based** box again to select and set all objects to rule based.

★ IMPORTANT

- The available questions, answers, and results vary depending upon the detailed parts associated with the connection.

- The questions in each **Selection** tab represent the default rules delivered with the software. Customized rules may have different questions.

Glossary

abaft

Toward the stern of a ship, behind, further aft than.

abstract part

A part that is only defined by a partial specification and that cannot be materially provided by the organization that defines the specification.

access holes

An opening cut in the structure of a ship to permit entering or leaving various compartments.

Active Template Library (ATL)

Set of class templates and wizards supplied with Microsoft C++ Version 5.0 and later. You can use an ATL when you create ActiveX controls and any other type of object that uses the Component Object Model (COM) model. Using an ATL is generally preferred over Microsoft Foundation Classes (MFC), because the implementations are smaller, easier to use, and more closely tied to the COM model.

aft

Toward, at, or near the stern.

after body

The hull from aft of the midship section.

aftermost

Nearest the stern.

angle

The circular measurement taken from the intersection of two pipes at a turn or branch.

approval state

Recorded state of acceptance of information contained in objects within the database. The approval states indicate a level of confidence in the information stored in the database and govern your ability to alter specific data about a product.

arrangement (accommodation)

Those components of a system arranged in three-dimensional space with accurate dimensional representation for installation. Various types include electrical, HVAC, machinery, outfitting, and piping.

attribute

A single type of non-graphics information that is stored about an object such as diameter or end preparation.

axis

An imaginary line used to define the orientation of a system or object normally defined in terms of an x-, y-, and z-axis. Some 3-D graphic objects have an associated axis used to define the center or axis for rotations.

basic design

Engineering definition of the model and its systems.

bending

Process of conforming a plate to the hull surface. A bending process can be either simple (bent in one direction) or compound (bent in multiple directions). The software must minimize compound bending requirements.

bill of material (BOM)

Hierarchical decomposition of a product into constituent assemblies and parts. Specific types of BOMs exist (for example, an EBOM is a bill of material from the point of view of an engineering department; an MBOM is a bill of material from the point of view of manufacturing).

brace

A diagonal member used to stiffen a framework.

built ships

Complete database of NGC information after completion of the ship contract.

built-up member

A member built from multiple standard shapes to create a single, usually stronger, member.

bulkload

The process by which reference data in Microsoft Excel workbooks is loaded into the Catalog database.

catalog

Repository of information about components and materials used in construction. When you use catalog parts in the model, the software places an occurrence of the catalog part in the project. This occurrence is a copy of the actual catalog part.

Catalog database

The database that contains the reference data. Each model database can reference a different Catalog database.

ceiling

Overhead design of the cabin area, including distribution systems for power, water, and ventilation.

chain

A set of continuous and tangent segments.

change history

Process of recording information such as who, when, and why for any given modification.

change management

Software features or manual procedures for managing the consequence of change. For example, software can support a change management feature to report drawings that need updating as a result of a change in a 3-D model.

change propagation

Ability of the software to intelligently modify dependent design information to reflect change in a higher order object.

class

Grouping of individual objects that share some very significant, common characteristics.

class rule check

Verification that the developing design meets the rules of a particular classification society, such as ABS, Lloyd's, or DNV.

Class Rules

Classification Society Design Rules.

classification folder

A folder in the Catalog hierarchy that contains part classes. Classification folders are one level above part classes. The ClassNodeType and R-ClassNodeDescribes sheets in the Microsoft Excel workbooks define the classification folders.

clip angle

A small angle-shaped piece of steel used for fastening members together.

codelist

A set of acceptable values for a particular property that can be referred to by an index number or selected in a combo box. For example, the codelist for the material specification allows you to select from a set of standard entries, such as ASTM A183-F316 Stainless Steel.

commodity code

A user-defined code that provides an index to parts in a catalog.

commodity item

A standard component found in a manufacturer catalog (an off-the-shelf component).

component

Physical part that a feature generates.

concurrent access

Ability of the software to allow multiple users to simultaneously access and modify the design of a model.

consolidated tasks

A collection of tasks run in batch. For example, the software allows you to extract a set of drawings immediately or to schedule the batch extraction for a future time.

constraints

A logical restriction that controls how part symbols ports relate to each other and to reference ports. There are four constraints: parallel, perpendicular, coincident, and distance.

contract

A Work Breakdown Structure object representing a scope of work, usually performed by an external supplier. The contract is related to a project and appears in the Work Breakdown Structure hierarchy.

control point

A point that is used to control the shape of a NURBS curve or surface. Curves have a one-dimensional array of control points, while surfaces have a two-dimensional array.

coordinate

The location of a point along the X-, Y-, or Z-axis.

coordinate system

A geometric relation used to denote the location of points in the model. The most common coordinate system is the rectangular coordinate system, whereby points are located by traversing the X-, Y-, and Z-axes of the model. Normally, coordinate systems have their origin defined as 0,0,0.

cope

To cut out the top or bottom flanges and possibly the web so that one member frames into another.

cover plate

A plate used in building up flanges, in a built-up member, to give greater strength and area, or for protection.

cross section

The shape of a member when viewed along the member line.

cutback

An axial offset that typically represents the distance from a member centerline to its face. Cutbacks are used to account for the difference between how a structure is modeled and constructed in terms of lengths of members and quantities of materials.

cutting plane

A plane that cuts through an object.

damage records

Data relating to the damage and repair of structure or components that occurred during or after construction of a plant.

data interchange

Capability to output the design, or portions of the design, in a standard format for use or movement to another computer software system.

database

Repository for the product model data. The database contains information to describe individual objects in the data model and the relationships between objects as appropriate.

database backup

Process of recording a backup copy of the complete database or the incremental changes after the date that the last complete copy was created.

database break and recovery

Utilities used to restore a database after files are corrupted.

database copy

Functionality to copy large collections of model objects from one design project to another design project.

database management

Functionality related to managing a product model database.

database monitor record

Transactions that occur in order to provide database (DB) recovery after a stop in response with a minimum of lost data.

degree

The highest polynomial factor in the curve or surface mathematical definition. A line is a degree 1 curve, while a cubic B-spline is a degree 3 curve.

degree of freedom

An allowable direction of movement, either translation or rotation. There are six possible degrees of freedom (DOFs): translation X, Y, and Z, and rotation RX, RY, and RZ.

design alternative

Difference in a design represented by a separate version. A design alternative can be a new design prepared as a proposed change, or one of several elective options that the builder or customer selects. Each design alternative has an identification assigned so you can uniquely refer to the design alternatives.

design approval log

Record of review and approval of parts of the design.

design data auto input

Automation in loading existing design data into a new design database.

design documents

Drawings, sketches, material lists, procedures, and so forth that are generated during the design phase.

design object

Any object with properties that you can select. A design object can be related to one or more contracts of different types, but related only to one contract of a given type.

design progress check

Analysis of the content of the design to some metric unit that gives an idea of the degree of completion.

design review

Functionality to support rapid viewing of the design and markup of features with comments.

design service

Any general system services related to the design function.

design standard

Feature or object used in plant design that has been determined to the normal or approved way of accomplishing a design requirement. In the context of computer software, the term refers to computer functionality to support standards, not the standard itself.

detail schedule

Lowest level of schedule used to manage and track work progress.

distributed systems

Systems consisting of sequential parts with a distributive characteristic (for example, pipes distribute fluids, HVAC distributes air, cabling distributes power, and structure distributes loads).

distribution systems

Term synonymous and used interchangeably with the term distributed systems.

documentation

Drawings and other records that you must produce to document, obtain approval, or build the design.

drawing tool

Tool that helps in the process of creating, modifying, or manipulating objects. Examples are PinPoint and SmartSketch.

easting

A term that describes an east coordinate location in a coordinate system.

edge

A topological object that represents a trimmed curve bounded by a start and end vertex.

edge distance

The distance from the center of a bolt or rivet to the edge of a plate or flange.

equipment catalog

Catalog of equipment geometry and limited properties that the software uses to identify and visualize equipment and its placement in the model. The catalog is not the source for the total specification and ordering data for the object.

external appendages

External structure attached to the hull, such as the propeller nozzle, shaft struts, bilge keel, and so forth.

fabricate

To cut, punch, and sub-assemble members in the shop.

face

A topological object that represents a trimmed surface bounded by a loop of edges.

face plate

An edge reinforcement type that places a plate or profile at the selected plate edge.

face-to-face

The overall length of a component from the inlet face to the outlet face.

fasteners

Bolts and rivets used to connect structural members.

element

Primitive geometric shape such as a line, circle, or arc.

fence

Boundary or barrier that separates or closes off an area. To surround or close like a fence.

field adjustment

Material added to the neat design geometry of piping or structural parts to allow for fit up in the case that extra material is required due to uncontrolled variance in the manufacturing and construction process.

fire integrity

Deck and bulkhead treatments and fire and smoke blocks for fire control and retardation.

flange

The projecting portion of a beam, channel, or column.

flavor

A different variation of a symbol. Each variation has different occurrence property values.

focus of rotation

A point or line about which an object or view turns.

full penetration weld

A type of weld in which the weld material extends through the complete thickness of the components being joined.

function points

Part of the requirements documentation, function points are the smallest granularity of a requirement statement that describe specific detailed actions that the software performs.

functional block diagram

Schematic representation of a system (piping, electrical, ventilation) showing system parts and their relationship. You use symbols to represent equipment and components. A connecting network of lines illustrates their relationship. Taken together, the symbols and the network illustrate the function of the system.

furnishings

Parts such as movable articles and fittings that normally are not associated with a system (for example, a chair).

generic specific

Object that is parametrically defined or defined to suit a family of specific parts (for example, International Standards parametrics). For example, a 100 - 200 gpm pump in the catalog can provide a general shape to appear in the model until a specific object has been identified. See also specific and specific object.

GUIDs

Acronym that stands for Globally Unique Identifiers. The software automatically creates the GUIDs sheet in the Excel workbooks when you create the Catalog database and schema. The purpose of storing GUIDs within Excel workbooks is to help you keep track of what has been loaded into the database. Storing GUIDs also helps to avoid the situation in which a replacement Catalog database causes existing models to become invalid.

host location

The first location created for a Site. This host location is defined when the Database Wizard creates the Site database.

host server

The database server on which the Site database was created using the Database Wizard. Alternatively, if it is a restored database set, the Host Server is the database server where the Site database is restored. The Host Server in a Workshare environment contains the origin for the Site, Site Schema, Catalog, and Catalog Schema databases. Consequently, most Project Management and reference data work must take place at the Host.

initial design

Early stage of design work, generally before contract, used to estimate construction costs and provide a rough concept of the intended plant. Contains information relating to a plant created during its initial (concept) design period.

initial structural plan

Principal structural plan for the plant; also called a construction profile.

instantiation

Occurrence of a catalog object at a specific geometric location in the model.

interference checking

A process that identifies possible collisions or insufficient clearance between objects in the model.

jigs and fixtures

Design of, or data for, devices that position work or hold work in position for joining, transport, or erection.

job order

Industrial authorization for accomplishing work; synonymous with a work order.

joiner

Non-structural bulkheads, and trim and built-in furnishings.

kinematics analysis

Analysis of mechanical motion.

ksi

Kips per square inch.

leg length analysis

Preferred term is welding length analysis.

library

Resource of reference information that you can access in developing a plant design.

life cycle database

Information developed to assist in the maintenance and modernization of delivered plants.

link

Way to store information about another file in your document. You can update a link so that changes in the file appear in your document.

lintel

A horizontal member used to carry a wall over an opening.

load group

A grouping in which all components feature uniform load limits and stress safety characteristics. For example, if a pipe clamp from load group 5 has a maximum nominal load of 20kN, then so does a threaded rod from load group 5.

location

A Location is defined by three user-defined inputs: 1) a unique name, 2) a unique name rule ID, and 3) the server where the Site databases reside for that Location. A Location is defined and created when the Site database is created using the Database Wizard. Additional Locations can be created in the Project Management task. Each Location is a Site-level object, thus other Plants within the same Site collection can use the Locations when the Plants are configured for Workshare.

logical member

An object in the model used to represent the design topology.

machinery

Major pieces of equipment installed in a plant.

macro

A sequence of actions or commands that can be named and stored. When you run the macro, the software performs the actions or runs the commands. You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the other OLE-aware programming applications are Visual Basic for Applications, Visual C++, and so forth.

maintenance envelope

A rectangular box around the part for clearance during maintenance operations.

maintenance parts

Required material for depot or on-board repair or overhaul of equipment, as determined by engineering study. Generally at a level below the purchased construction object of the plant.

maintenance records

Records of breakdown, repair, and overhaul of equipment.

material analysis

Analysis of a completed design work for extracting detailed material requirements; also called material lists.

material list

An option category that controls the format and content of the bill of materials.

member name

A user-definable alphanumeric code used to uniquely identify individual members in the model.

member part

A model object derived from the logical model that represents the manufactured physical member parts.

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member system

A logical collection of member parts that can be moved as a single entity.

methods

Objects in the database that describe the manufacturing methods to the component parts of a plant.

move from point

Starting point for an action. For example, when you move an equipment object, the Move From point determines the point of origin for the move.

move to point

Ending point for an action. For example, when you move an equipment object, the Move To point determines where you want the move to stop.

MTO neutral file

A non-graphic output file that can be fed into a material control system. MTO stands for Material Take-Off.

natural surface

A surface without a boundary curve.

node

- One of the set of discrete points in a flow graph.
- A terminal of any branch of a network or a terminal common to two or more branches of a network.
- An end point of any branch or a network or graph, or a junction common to two or more branches.

northing

A term that describes a north coordinate location in a coordinate system.

nozzle

A piping connection point to a piece of equipment.

nozzle standout

The shortest allowable distance between the connection point of a nozzle and the start point of a turn on the leg connected to the nozzle.

NPD (Nominal Piping Diameter)

The diameter of a pipe.

object

A type of data other than the native graphic format of the application.

occurrence (of part or equipment)

Instantiation of a part of equipment in the model that refers to the part library; an instance of a specific object. The design can be built several times, and therefore the occurrence can apply to more than one hull. Typically, an occurrence points back to a specific object, either for its complete definition, as in the case of a particular valve, or for its made from material, as in the case of a steel plate part cut from sheets. Thus, when a designer selects a component from the catalog and places it at a location in the space of the plant, the software creates an occurrence of that object in the plant design.

occurrence property

A characteristic that applies to an individual object in the model. Occurrence properties are designated with 'oa:' in the reference data workbooks. You can view and modify occurrence properties on the Occurrence tab of the properties dialog boxes in the software. Depending on the object, some occurrence properties are read-only.

origin

In coordinate geometry, the point where the X-, Y-, and Z-axes intersect.

origin point

The point at which the coordinate system is placed, providing a full Cartesian coordinate system with positive and negative quadrants. Points are placed at coordinates relative to the origin point, represented by the X, Y, and Z values.

orthogonal

The characteristic of an element consisting completely of elements positioned at 90-degree angles. A square is an orthogonal element.

orthographic

A depiction of an object created by projecting its features onto a plane along lines perpendicular to the plane.

P&ID

Diagram that shows the topology, functional components, and special requirements of a piping system; generally represents the engineering design of the system.

package

Set of closely related classes. (UML)

painting

Computation of paint surface and recording of paint system requirements.

parameter

A property whose value determines the characteristics or behavior of something.

part class

A group of similar objects. You can define part classes in the Excel workbooks. A part class can have multiple parts. For example, a heat exchanger part class can contain heat exchangers with different dimensions.

part number

Unique identifier of a part.

PDS (Plant Design System)

A comprehensive, intelligent, computer-aided design and engineering application for the process, power, and marine industries. PDS consists of integrated 2-D and 3-D modules that correspond to engineering tasks in the design workflow.

physical occurrence

Unique specific object that has traceability and is the physical manifestation of an occurrence object. A physical occurrence applies to one and only one hull. It is a version of its occurrence object with as-built or as-modified differences included and has a serial number or lot number.

PinPoint

Tool that allows you to place, move, and modify elements with precision, relative to a reference point.

plate

A flat, rectangular steel shape.

principle of superposition

The principle that states that the stresses, strains, and displacements due to different forces can be combined. This principle is only valid for linear analysis.

product

Data objects that describe the components of a ship and any corresponding properties. An individual object or part (or its representation in the product model) that may be installed in the ship. Examples of individual products include objects such as a coffee urn, a light fixture, a piece of pipe, a piece of ventilation duct, a radar display console, a bulkhead plate, and a structural profile stiffening a bulkhead.

Product Data Management (PDM) System

Software intended to manage both product data and documents associated to the product data. Functionality typically includes: object-based data modeling tools, user administration, business rules, and document management. Document management typically includes document editing or reviewing, document mark-up or redline, document storage, and full-text retrieval.

product structure

Hierarchical breakdown or decomposition of a product into constituent parts, volumes, or units. (For example, a bill of material is one possible type of product structure.)

production planning

Functionality associated with the work breakdown and sequence of the construction of a plant.

promotion

Process of associating approval state with a product version. A product version begins its existence at a working approval state. When the version is at some level of maturity, its approval state is elevated to a higher approval state (that is, promoted). Then, further changes must be

carefully controlled and generally require the data set demoted to a working state. One or more promotions can occur successively higher approval states (between working and approved) to represent various intermediate levels of review or progressive approval.

query select sets

Set of objects that are selected in a query or queries on the database.

reference data

The data that is necessary to design plants or ships using the software. Reference data includes graphical information, such as symbols. It also contains tabular information, such as physical dimensions and piping specifications.

resource estimation

Rough estimate of material, manpower, and facility utilization for the design and construction of the plant.

route

1) A line connecting a series of points in space and constituting a proposed or traveled route. 2) The set of links and junctions joined in series to establish a connection.

satellite server

The database server where the replicated databases reside for Workshare. The Satellite Server is not used unless Workshare is activated.

schema

A database that creates the structure of another database. For example, a schema specifies the queries, tables, fields, and data types in a database.

schema update utility

Functionality used to assist in processing existing product models to an updated database structure after you modify or add to the database structure.

sheetbody

A topological object that represents a collection of faces joined along their common edges (stitched).

shell structure

External portion of the surface of the plant.

ship

A collection of modeled objects that can be simultaneously displayed and edited in a workspace. A Ship points to a Catalog (optionally shared with other Ships). Access control is managed at the Ship level.

site

The top level in the Project Management hierarchy. A Site configuration may contain several Catalogs, each shared by multiple Plants.

site administrator

Person responsible for managing the standards and general parameters for a given plant site within a Site database.

site setup

Functionality associated with establishing a new plant site or hull for design development.

sketch and trace

User interface for rough definition of a required design feature that typically works in a 2-D mode.

specials

An option category that allows you to control specialized calculations for equipment trim, repeatability, and center-of-gravity.

specifications

Contracted requirements for the plant.

steel outfitting

Internal structural elements of a ship that are required to meet a local requirement such as foundations, non-structural bulkheads, walkways, and so forth.

stern frame

Casting and structure that support the rudder and shaft opening.

stiffener

An angle, plate, or channel fastened to a member to prevent buckling.

stud

A bolt, threaded on both ends, used to connect components.

suspended floor

A concrete floor system built above and off the ground.

swash bulkhead

A longitudinal or transverse nontight bulkhead in a tank that decreases the swashing motion of the liquid contents. A plate in a tank that has this same effect but that does not extend to the bottom of the tank is called a swash plate.

symmetric node

Type of vertex on a curve. A curve with a symmetric node has the same curvature on each side of the node. A handle can be attached to a symmetric node for editing.

system

A conceptual design grouping that organizes parts in hierarchical relationships. A system represents a functional view of the model and includes information such as system name, type, properties, and design specifications for the objects assigned to the system.

tag number

User-specific, unique number assigned to an object (for example, CV-101 for a control valve, HE-2002 for a heat exchanger).

target point

The origin for coordinate measurements displayed by PinPoint. You can position the target point anywhere on the drawing sheet or view.

tolerant geometry

A type of ACIS geometry - either an edge or a vertex - that is outside the tolerance for ACIS and requires special handling.

transverse

At right angles to the fore-and-aft center line.

transverse frames

The athwartship members that form the ribs of the ship.

trim

The difference between the forward draft and the aft draft.

trimmed surface

A surface whose boundary is fully or partially inside the "natural" geometric definition of the surface. Some or the entire control polygon extends outside the face boundary.

trunk

Feature that quickly reserves space for the distributive systems and other systems that have a path. Along the trunk are stations that define the cross section and identify part or system membership.

tumble home

The inboard slope of the side of a ship, usually above the designed waterline.

unit/module modeler

Facility of the system to structure collections of equipment and components into a single identifiable object.

user attributes

A customized property in the reference data. The Custom Interfaces sheets in the Excel workbooks define these properties. You can list the customized properties on the individual part class sheets.

version control

Ability of the system to manage multiple versions of a single part of the design. Version control should support conditional analysis and promotion status, as well as alternate design features among hulls within a plant site.

vertex

A topological object that represents a point in the three-dimensional model.

vertical keel

A row of vertical plates extending along the center of the flat plate keel.

viewset

Set of objects (usually a subset of the entire database) that a view operation uses. Membership or lack of membership for any object in a viewset does not affect the actual stored representation of the object, but only its availability or desirability for viewing in the current scenario.

water line

A line parallel with the base line that depicts the water.

watertight door

A door that when closed prevents the passage of water.

weather deck

A deck exposed to the weather.

weathertight door

A door that when closed prevents the passage of rain and spray.

weight and CG analysis

Routines that compute the weight of commodity materials as configured in a given design (for example, plate and pipe) and determine total weight and center of gravity (CG) for a collection of material and equipment, as well as the complete plant.

welding

Weld requirements for joining materials. Welding length analysis is the calculation of required weld dimensions; also called leg length analysis.

windlass

The machine used to hoist and lower anchors.

wirebody

A topological object that represents a collection of edges jointed at their common endpoints.

wizard

Software routine attached to an application that provides guidance and expert help to you to complete one of the functionalities of the application.

work content

Estimation development of metrics from the database that relates to the work hour content of the various construction units.

work order

Plant authorization for completing work; synonymous with a job order.

working plane

The available 2-D plane of movement for endpoint selection.

workset

Set of objects (usually a subset of the entire database) used in an interactive change, add, or delete operation. Membership or lack of membership for any object in a workset does not necessarily affect the actual stored representation of an object. However, you can change or delete an object in a workset that also results in a change or deletion of the stored object. Similarly, when you add a new object (not currently stored) to a workset, the software also adds the object container.

workspace

Area that represents the portion of the model data needed to perform the intended task and includes the user modeling settings.

workspace document

Document into which you can extract a portion of the model data for a user task.

Workspace Explorer

Tree or list representation of objects in your workspace.

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